STRUCTURAL SYSTEMS

RESEARCH PROJECT

Report No.
SSRP–14/04

MSBRIDGE: OPENSEES PUSHOVER AND EARTHQUAKE ANALYSIS OF MULTI-SPAN BRIDGES - USER MANUAL

by

AHMED ELGAMAL

JINCHI LU

KEVIN MACKIE

Final Report Submitted to the California Department of Transportation (Caltrans) under Contract No. 65A0445.

May 2014

Department of Structural Engineering
University of California, San Diego
La Jolla, California 92093-0085

University of California, San Diego
Department of Structural Engineering
Structural Systems Research Project

Report No. SSRP-14/03

MSBridge: OpenSees Pushover and Earthquake Analysis of Multi-span Bridges - User Manual

by

Ahmed Elgamal
Professor of Geotechnical Engineering

Jinchi Lu
Assistant Project Scientist

Kevin Mackie
Associate Professor of Structural Engineering
at University of Central Florida

Final Report Submitted to the California Department of Transportation under Contract No. 65A0445

Department of Structural Engineering
University of California, San Diego
La Jolla, California 92093-0085

May 2014

Technical Report Documentation Page

1. Report No.	2. Government Accession No.	3. Recipient's Catalog No.

4. Title and Subtitle	5. Report Date
MSBridge: OpenSees Pushover and Earthquake Analysis of Multi-span Bridges - User Manual	May 2014
	6. Performing Organization Code

7. Author(s)	8. Performing Organization Report No.
Ahmed Elgamal and Jinchi Lu	UCSD / SSRP-14/04

9. Performing Organization Name and Address	10. Work Unit No. (TRAIS)
Department of Structural Engineering School of Engineering University of California, San Diego La Jolla, California 92093-0085	11. Contract or Grant No. 65A0445

12. Sponsoring Agency Name and Address	13. Type of Report and Period Covered
California Department of Transportation Division of Engineering Services 1801 30th St., MS-9-2/5i Sacramento, California 95816	Final Report
	14. Sponsoring Agency Code

15. Supplementary Notes

Prepared in cooperation with the State of California Department of Transportation.

16. Abstract

MSBridge is a PC-based graphical pre- and post-processor (user-interface) for conducting nonlinear Finite Element (FE) studies for multi-span multi-column bridge systems. Finite element computations are conducted using OpenSees (http://opensees.berkeley.edu), an open source framework developed by the Pacific Earthquake Engineering Research (PEER) Center. The analysis options available in MSBridge include: i) Pushover Analysis; ii) Mode Shape Analysis; iii) Single 3D Base Input Acceleration Analysis; iv) Multiple 3D Base Input Acceleration Analysis; and v) Equivalent Static Analysis (ESA). This document describes how to conduct the above analyses in MSBridge. For information about how to download and install MSBridge, please visit the MSBridge website (http://www.soilquake.net/msbridge/).

17. Key Words	18. Distribution Statement
Finite Element, Time History Analysis	Unlimited

19. Security Classification (of this report)	20. Security Classification (of this page)	21. No. of Pages	22. Price
Unclassified	Unclassified		

Form DOT F 1700.7 (8-72) Reproduction of completed page authorized

DISCLAIMER

This document is disseminated in the interest of information exchange. The contents of this report reflect the views of the authors who are responsible for the facts and accuracy of the data presented herein. The contents do not necessarily reflect the official views or policies of the State of California or the Federal Highway Administration. This publication does not constitute a standard, specification or regulation. This report does not constitute an endorsement by the California Department of Transportation of any product described herein.

For individuals with sensory disabilities, this document is available in Braille, large print, audiocassette, or compact disk. To obtain a copy of this document in one of these alternate formats, please contact: the Division of Research and Innovation, MS-83, California Department of Transportation, P.O. Box 942873, Sacramento, CA 94273-0001.

ACKNOWLEDGMENTS

The research described in this report was supported by the California Department of Transportation (Caltrans) under Contract No. 65A0445. This support is most appreciated.

ABSTRACT

MSBridge is a PC-based graphical pre- and post-processor (user-interface) for conducting nonlinear Finite Element (FE) studies for multi-span multi-column bridge systems. Finite element computations are conducted using OpenSees (http://opensees.berkeley.edu), an open source framework developed by the Pacific Earthquake Engineering Research (PEER) Center. The analysis options available in MSBridge include: i) Pushover Analysis; ii) Mode Shape Analysis; iii) Single 3D Base Input Acceleration Analysis; iv) Multiple 3D Base Input Acceleration Analysis; and v) Equivalent Static Analysis (ESA). This document describes how to conduct the above analyses in MSBridge. For information about how to download and install MSBridge, please visit the MSBridge website (http://www.soilquake.net/msbridge/).

TABLE OF CONTENTS

DISCLAIMER ..... **iv**
ACKNOWLEDGMENTS ..... **v**
ABSTRACT ..... **vi**
TABLE OF CONTENTS ..... **vii**
LIST OF FIGURES ..... **xi**
LIST OF TABLES ..... **xvii**
1 Introduction **18**
1.1 Overview 18
1.1 Units 18
1.2 Coordinate Systems 19
1.3 System Requirements 20
1.4 Acknowledgments 20
2 Getting Started **21**
2.1 Start-Up 21
2.2 Interface 21
2.2.1 Menu Bar 22
2.2.2 Model Input Region 25
2.2.3 Finite Element Mesh Region 25
3 Bridge Model **27**
3.1 Spans 28
3.1.1 Straight Bridge 28
3.1.2 Curved Bridge 29
3.1.2.1 Horizontal Curved Bridge 29
3.1.2.2 Vertical Curved Bridge 30
3.2 Deck 34
3.3 Bentcap 35
3.4 Columns 37
3.4.1 Column Heights 38

3.4.2 Column Connection 38
3.4.3 Column Properties 39
3.5 Foundation 52
3.5.1 Rigid Base 52
3.5.2 Soil Springs 52
3.5.3 Foundation Matrix 59
3.6 Advanced Options 60
3.6.1 Deck Hinges 61
3.6.2 Isolation Bearings 64
3.6.3 Steel Jackets 66
3.6.4 Skew Angles 67
3.7 Mesh Parameters 69
4 Abutment Models 70
4.1 Elastic Abutment Model 70
4.2 Roller Abutment Model 75
4.3 SDC 2004 Abutment Model 76
4.3.1 Longitudinal Response 76
4.3.2 Transverse Response 80
4.3.3 Vertical Response 82
4.3.4 Definition of the SDC 2004 Abutment Model 84
4.4 SDC 2010 Sand Abutment Model 87
4.5 SDC 2010 Clay Abutment Model 88
4.6 ElasticPP-Gap Model 89
4.7 HFD Model 90
5 Column Responses & Bridge Resonance 93
5.1 Bridge Natural Periods 93
5.2 Column Gravity Response 93
5.3 Column & Abutment Longitudinal Responses 93
5.4 Column & Abutment Transverse Responses 93
6 Pushover & Eigenvalue Analyses 97
6.1 Monotonic Pushover 97

6.2 Cyclic Pushover 98
6.3 User-Defined Pushover (U-Push) 99
6.4 Output for Pushover Analysis 100
6.4.1 Column Response Profiles 100
6.4.2 Column Response Time Histories 101
6.4.3 Column Response Relationships 103
6.4.4 Abutment Force-Displacement and Response Time Histories 104
6.4.5 Deformed Mesh 105
6.5 Eigenvalue Analysis 106
7 Ground Shaking 112
7.1 Definition/specification of input motion ensemble (suite) 112
7.1.1 Available Ground Motions 112
7.1.2 Specifications of Input Motions 113
7.1.3 Rayleigh Damping 120
7.2 Save Model and Run Analysis 121
8 Time History and Engineering Demanding Parameter Output 125
8.1 Time History Output Quantities 125
8.1.1 Deck Response Time Histories 126
8.1.2 Column Response Profiles 126
8.1.3 Column Response Time Histories 126
8.1.4 Column Response Relationships 129
8.1.5 Abutment Responses Time Histories 131
8.1.6 Soil Spring Responses Time Histories 134
8.1.7 Deck Hinge Responses Time Histories 135
8.1.8 Isolation Bearing Responses Time Histories 137
8.2 Deformed Mesh and Animation 138
8.3 Maximum Output Quantities 141
8.3.1 EDP Quantities 141
8.3.2 Bridge Peak Accelerations & Displacements for All Motions 143
8.3.3 Maximum Column & Abutment Forces for All Motions 145
9 Equivalent Static Analysis 147

9.1 Bridge Longitudinal Direction 147
9.2 Bridge Transverse Direction 147
Appendix A Calculation of Steel and Concrete Material Properties.....153
Appendix B How to Incorporate User-defined Motions.....158
Appendix C Comparison with SAP2000 for Representative OB Configurations 163
References 168

LIST OF FIGURES

Fig. 1. Global coordinate system employed in **MSBridge** 19
Fig. 2. MSBridge main window 21
Fig. 3. **MSBridge's** menu and submenu bars: a) menu bar; b) menu **File**; c) menu **Execute**; d) menu **Display**; e) menu **Report**; and f) menu **Help** 23
Fig. 4. MSBridge copyright and acknowledgment window 25
Fig. 5. Available actions in the FE Mesh window 26
Fig. 6. Model builder buttons 27
Fig. 7. MSBridge main window (bridge model with soil springs and deck hinges included) 27
Fig. 8. Spans 28
Fig. 9. Varied span lengths 29
Fig. 10. Straight bridge with different span lengths 29
Fig. 11. Horizontal alignment 30
Fig. 12. Horizontal and vertical alignments: a) horizontal alignment (plan view); b) vertical alignment (side view) 31
Fig. 13. Vertical alignment 31
Fig. 14. Examples of horizontal curved bridges (horizontal alignment): a) single radius horizontal curve; b) multi-radius horizontal curve; c) horizontal curve connecting to straight parts). 33
Fig. 15. Examples of vertically curved bridges (vertical alignment): a) single slope; b) begin and end slopes; c) multiple slopes; d) mixing slope and zero-slope. 33
Fig. 16. Material properties of the bridge deck 34
Fig. 17. Box girder shape employed for the bridge deck 35
Fig. 18. Material properties of the bent cap 36
Fig. 19. Rectangular shape employed for the bent cap 37
Fig. 20. Columns 38
Fig. 21. Column heights 38
Fig. 22. Column boundary conditions 39

Fig. 23. Column properties and available beam-column element types 41
Fig. 24. Definition of linear column 42
Fig. 25. Column Elastic material properties 42
Fig. 26. Column Section properties 42
Fig. 27. Nonlinear Fiber Section window 43
Fig. 28. Column nonlinear material properties: a) Steel02 material; b) ReinforcingSteel material; c) Concrete02 material for the core concrete; d) Concrete02 material for the cover material 45
Fig. 29. Column fiber section (based on PEER best modeling practices report, Berry and Eberhard, 2007): a) Circle; b) Octagon; c) Hexagon 46
Fig. 30. OpenSees *quadrilateral patch* employed for calculating the cover concrete fibers for: a) Octagon; and b) Hexagon cross section 46
Fig. 31. Stress-strain curve for a steel material (default values employed): a) Steel01 (with a strain limit); b) Steel02 (with a strain limit); and c) ReinforcingSteel (with a strain limit) 48
Fig. 32. Stress-strain curve of the core concrete material (default values employed): a) Elastic-No Tension; b) Concrete01; and c) Concrete02 50
Fig. 33. Stress-strain curve of the cover concrete material (default values employed): a) Concrete01; and b) Concrete02 51
Fig. 34. Moment-curvature response for the column (with default steel and concrete parameters, and the deck weight 2680 kip applied at the column top 51
Fig. 35. Foundation types available in **MSBridge** 52
Fig. 36. Shaft foundation for abutments and bents 53
Fig. 37. Pile foundation model for abutments 54
Fig. 38. Soil springs 55
Fig. 39. Soil spring calculations based on p-y equations: a) Soft Clay (Matlock); b) Stiff Clay without Free Water (Reese); c) Sand (Reese) 57
Fig. 40. Soil spring definition window after using the soil spring data calculated based on p-y equations 58
Fig. 41. Sample FE mesh of a bridge model with soil springs included 58
Fig. 42. Local coordination system for the foundation matrix 59

Fig. 43. Foundation matrix for each bent 60
Fig. 44. Advanced options 61
Fig. 45. Definition of deck hinges 63
Fig. 46. FE mesh of a 4-span model with 2 deck hinges included 63
Fig. 47. OpenSees zeroLength elements for deck hinges (plan view) 64
Fig. 48. Definition of isolation bearings 65
Fig. 49. FE mesh of a 4-span bridge model with 2 isolation bearings included on each bent cap 65
Fig. 50. OpenSees zeroLength elements for isolation bearings (side view of bent cap cut-plan) 66
Fig. 51. Definition of steel jackets 67
Fig. 52. Sketch of steel jacket 67
Fig. 53. Definition of skew angles 68
Fig. 54. Mesh parameters 69
Fig. 55. Definition of an abutment model 71
Fig. 56. General scheme of the Elastic Abutment Model: a) longitudinal component; b) transverse component; c) vertical component 72
Fig. 57. Definition of the Elastic Abutment Model 73
Fig. 58. Parameters of the Elastic Abutment Model 73
Fig. 59. Longitudinal components of the Elastic Abutment Model in a curved bridge: a) left abutment; b) right abutment 74
Fig. 60. Bridge model with multiple distributed springs and a positive skew angle: a) straight bridge; b) curved bridge 75
Fig. 61. General scheme of the Roller Abutment Model 76
Fig. 62. Selection of the Roller Abutment Model 76
Fig. 63. Longitudinal response of the SDC 2004 Abutment Model: a) general scheme; b) longitudinal response of a bearing pad; c) total longitudinal response 79
Fig. 64. Longitudinal response of the SDC 2004 Abutment Model for a curved bridge with a (positive) skew angle 80

Fig. 65. Transverse response of the SDC 2004 Abutment Model: a) general scheme; b) response of a bearing pad and shear keys (curve with a higher peak value is the shear key response); c) total transverse response 82
Fig. 66. Vertical response of the SDC 2004 Abutment Model: a) general scheme; b) vertical response of a bearing pad; c) total vertical response..... 84
Fig. 67. Definition of the SDC 2004 Abutment Model: a) main parameters; b) bearing pad properties; c) shear key properties; d) SDC abutment properties; e) embankment properties 87
Fig. 68. Backfill horizontal properties for the SDC 2010 Sand Abutment Model 88
Fig. 69. Backfill horizontal properties of the SDC 2010 Clay Abutment Model 89
Fig. 70. Backfill horizontal properties of the EPP-Gap Abutment Model..... 89
Fig. 71. Definition of the HFD Abutment Model: a) HFD abutment model; and b) HFD parameters for abutment backfills suggested by Shamsabadi et al. (2007); and c) backfill properties of the HFD Model 92
Fig. 72. Buttons to view column & abutment responses and bridge resonance 93
Fig. 73. Natural periods and frequencies of bridge..... 94
Fig. 74. Column internal forces and bending moments after application of own weight. 94
Fig. 75. Column longitudinal responses 95
Fig. 76. Abutment longitudinal responses 95
Fig. 77. Column transverse responses..... 96
Fig. 78. Abutment transverse responses 96
Fig. 79. Pushover analysis option 97
Fig. 80. Load pattern for monotonic pushover analysis..... 98
Fig. 81. Load pattern for cyclic pushover analysis 99
Fig. 82. User-defined pushover (U-Push) 100
Fig. 83. Column response profiles 101
Fig. 84. Column response time histories..... 102
Fig. 85. Response relationships for column..... 103
Fig. 86. Abutment response time histories..... 104
Fig. 87. Deformed mesh and contour fill 105
Fig. 88. Visualization of Plastic hinges (red dots represent plastic hinges developed) .. 106

Fig. 89. Steps to perform an Eigenvalue analysis 107
Fig. 90. Sample output for an Eigenvalue analysis for the default bridge model: a) first mode; b) second mode; c) third mode; d) fourth mode; and e) fifth mode 111
Fig. 91. Group shaking analysis 114
Fig. 92. Input motions window 115
Fig. 93. Importing a user-defined motion: a) choosing data files; b) message showing new motion has been added 117
Fig. 94. Time histories and response spectra of individual motion 118
Fig. 95. Intensity measures of individual motion 118
Fig. 96. Histogram and cumulative distribution for the whole input motion set: a) histogram; b) cumulative distribution 120
Fig. 97. Rayleigh damping 122
Fig. 98. Simultaneous execution of analyses for multiple motions 123
Fig. 99. Parameters for OpenSees analysis 124
Fig. 100. Selection of an input motion 125
Fig. 101. Deck longitudinal displacement response time histories 127
Fig. 102. Displacement profile in the longitudinal plane 127
Fig. 103. Bending moment profile in the longitudinal plane 128
Fig. 104. Response time histories and profiles for column (and pile shaft): displacement is shown at the nodes. 129
Fig. 105. Load-displacement curve at column top 130
Fig. 106. Moment-curvature curve at column top 131
Fig. 107. Abutment response force-displacement relationships: a) longitudinal response; b) transverse response; and c) vertical response 134
Fig. 108. Soil spring response time histories 135
Fig. 109. Deck hinge response time histories: a) cable element; b) edge element 137
Fig. 110. Isolation bearing response time histories 138
Fig. 111. Deformed mesh 139
Fig. 112. Visualization of Plastic Hinges 140
Fig. 113. EDP quantities for all motions 143

Fig. 114. Bridge peak accelerations for all motions: a) maximum bridge accelerations; b) maximum bridge displacements 145
Fig. 115. Maximum column & abutment forces for all motions 146
Fig. 116. Equivalent Static Analysis for the bridge longitudinal & transverse directions148
Fig. 117. Sample output of ESA for the bridge longitudinal direction: a) pushover load; b) elastic displacement demand 149
Fig. 118. Comparison of displacements from ESA and THA 150
Fig. 119. Output of ESA for the bridge transverse direction: a) pushover load and bent number; b) elastic displacement demand 152
Fig. 120. Choosing a motion set 160
Fig. 121. Directory structure of a motion set 161
Fig. 122. Sample .info file 161
Fig. 123. Sample .data file 162
Fig. 124. Example of user-defined motion 162
Fig. 125. Bridge Type 1 model: a) **MSBridge**; b) SAP2000..... 164
Fig. 126. Bridge Type 2 model: a) **MSBridge**; b) SAP2000..... 165
Fig. 127. Bridge Type 9 model: a) **MSBridge**; b) SAP2000..... 166
Fig. 128. Bridge Type 10 model: a) **MSBridge**; b) SAP2000..... 167

LIST OF TABLES

Table 1. Default values for column reinforced concrete (RC) section properties 46
Table 2. Default values for Steel02 material properties 47
Table 3. Default values for Concrete02 material properties 47
Table 4. Geometric and Material Properties of a Bearing Pad 77
Table 5. SDC Abutment Properties 88
Table 6. Engineering Demand Parameters (EDP) 141
Table 7. Typical Bridge Configurations in California (After Ketchum et al 2004) 163
Table 8. Displacement (unit: inch) of Bridge Type 1 under pushover (load of 2000 kips applied at deck center along both the longitudinal and transverse directions) 164
Table 9. Displacement (unit: inch) of Bridge Type 2 under pushover (load of 2000 kips applied at deck center along both the longitudinal and transverse directions) 165
Table 10. Displacement (unit: inch) of Bridge Type 9 under pushover (load of 1000 kips applied at deck center along both the longitudinal and transverse directions) 166
Table 11. Displacement (unit: inch) of Bridge Type 10 under pushover (load of 2000 kips applied at deck center along both the longitudinal and transverse directions) 167

1 Introduction

1.1 Overview

MSBridge is a PC-based graphical pre- and post-processor (user-interface) for conducting nonlinear Finite Element (FE) studies for multi-span multi-column bridge systems. Main features include:

i) Automatic mesh generation of multi-span (straight or curved) bridge systems
ii) Options of foundation soil springs and foundation matrix
iii) Options of deck hinges, isolation bearings, and steel jackets
iv) Management of ground motion suites
v) Simultaneous execution of nonlinear time history analyses for multiple motions
vi) Visualization and animation of response time histories

Finite element computations are conducted using OpenSees (http://opensees.berkeley.edu, McKenna et al. 2010, Mazzoni et al. 2009), an open source framework developed by the Pacific Earthquake Engineering Research (PEER) Center. The analysis options available in **MSBridge** include:

i) Pushover Analysis
ii) Mode Shape Analysis
iii) Single 3D Base Input Acceleration Analysis
iv) Multiple 3D Base Input Acceleration Analysis
v) Equivalent Static Analysis (ESA)

This document describes how to conduct the above analyses in **MSBridge**. For information on how to download and install **MSBridge**, please visit the **MSBridge** website (http://www.soilquake.net/msbridge/).

1.1 Units

MSBridge supports analysis in both the US/English and SI unit systems, the default system is US/English units. This option is located at the top of the main window (Fig. 2), and can be interchanged during model creation, **MSBridge** will convert all input data to the new unit system.

For conversion between SI and English Units, please check:
http://www.unit-conversion.info/

Some commonly used quantities can be converted as follows:

1 kPa = 0.14503789 psi
1 psi = 6.89475 kPa
1 m = 39.37 in

1 in = 0.0254 m

1.2 Coordinate Systems

The global coordinate system employed in **MSBridge** is shown in Fig. 1. The origin is located at the left deck-end of the bridge. The bridge deck direction in a straight bridge is referred to as "longitudinal direction (X)", while the horizontal direction perpendicular to the longitudinal direction is referred to as "transverse direction (Y)". In a curved bridge, the bridge deck direction at the left deck-end will be used as the longitudinal direction. At any time, "Z" denotes the vertical direction.

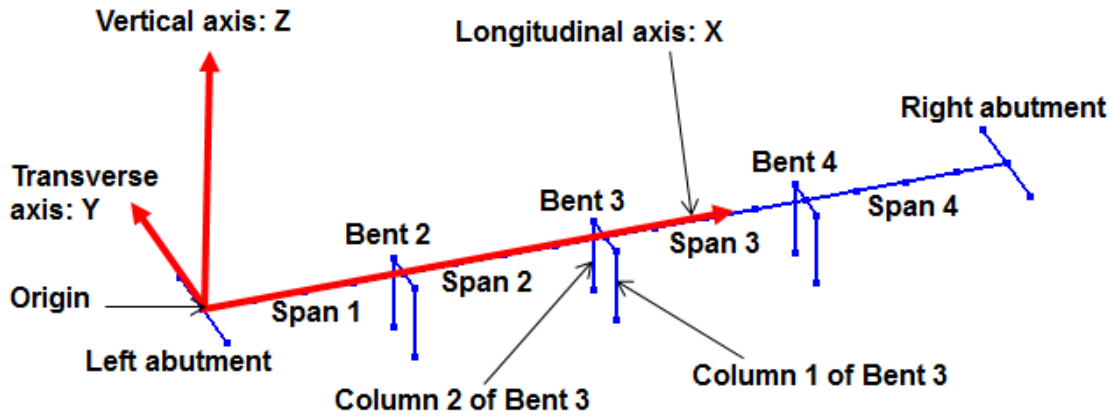


Fig. 1. Global coordinate system employed in **MSBridge**

When referencing different members and locations, the numbering and names used in **MSBridge** follow designations as follows: The left abutment is designated "Abutment 1" or "Left Abutment". Moving rightward, and starting with Bent 2, the bents are numbered consecutively. The right abutment is designated "Right Abutment" or "Abutment *N*" (where *N* is the last Bent number plus one, e.g., the right abutment can be referred to as "Abutment 5"). The span numbering corresponds to the abutment and bent numbering, so, Span 1 goes from Abutment 1 to Bent 2, and so on.

For multi-column scenarios, the columns are numbered consecutively along the transverse (Y) direction, starting from 1 in the most negative side. e.g., in Fig. 1, the columns at the negative side of the transverse (Y) direction are referred to as Column 1 while those at the positive side are called Column 2. For Bent 3, there are "Column 1 of Bent 3" and "Column 2 of Bent 3", which are used in **MSBridge** when referencing these 2 columns.

Local coordinate systems will also be used in this document to describe certain components, e.g., deck hinges, isolation bearings, distributed spring abutment models with a skew angle, etc. In that case, labels of "1", "2" and "3" (or lower case "x", "y" and "z") will be used. Please refer to appropriate section for the corresponding description.

1.3 System Requirements

MSBridge runs on PC-compatible systems using Windows (NT V4.0, 2000, XP, Vista or 7 & 8). The system should have a minimum hardware configuration appropriate to the particular operating system. For best results, the system's video should be set to 1024 by 768 or higher.

1.4 Acknowledgments

This research project was funded by California Department of Transportation (CalTrans).

OpenSees (currently ver. 2.4.0 is employed) is a software framework (McKenna et al. 2010) for developing applications to simulate the performance of structural and geotechnical systems subjected to earthquakes (for more information, please visit http://opensees.berkeley.edu/).

MSBridge is written in Microsoft .NET Framework (Windows Presentation Foundation or WPF). OpenTK (OpenGL) library (website: http://www.opentk.com/) is used for visualization of FE mesh. And OxyPlot (http://oxyplot.codeplex.com/) is employed for x-y plotting.

For questions or remarks about **MSBridge**, please send email to Dr. Ahmed Elgamal (elgamal@ucsd.edu), or Dr. Jinchi Lu (jinlu@ucsd.edu).

2 Getting Started

2.1 Start-Up

On Windows, start **MSBridge** from the Start button or from an icon on your desktop. To Start **MSBridge** from the Start button:

i) Click Start, and then select All Programs.
ii) Select the **MSBridge** folder
iii) Click on **MSBridge** (icon:)

The **MSBridge** main window is shown in Fig. 2.

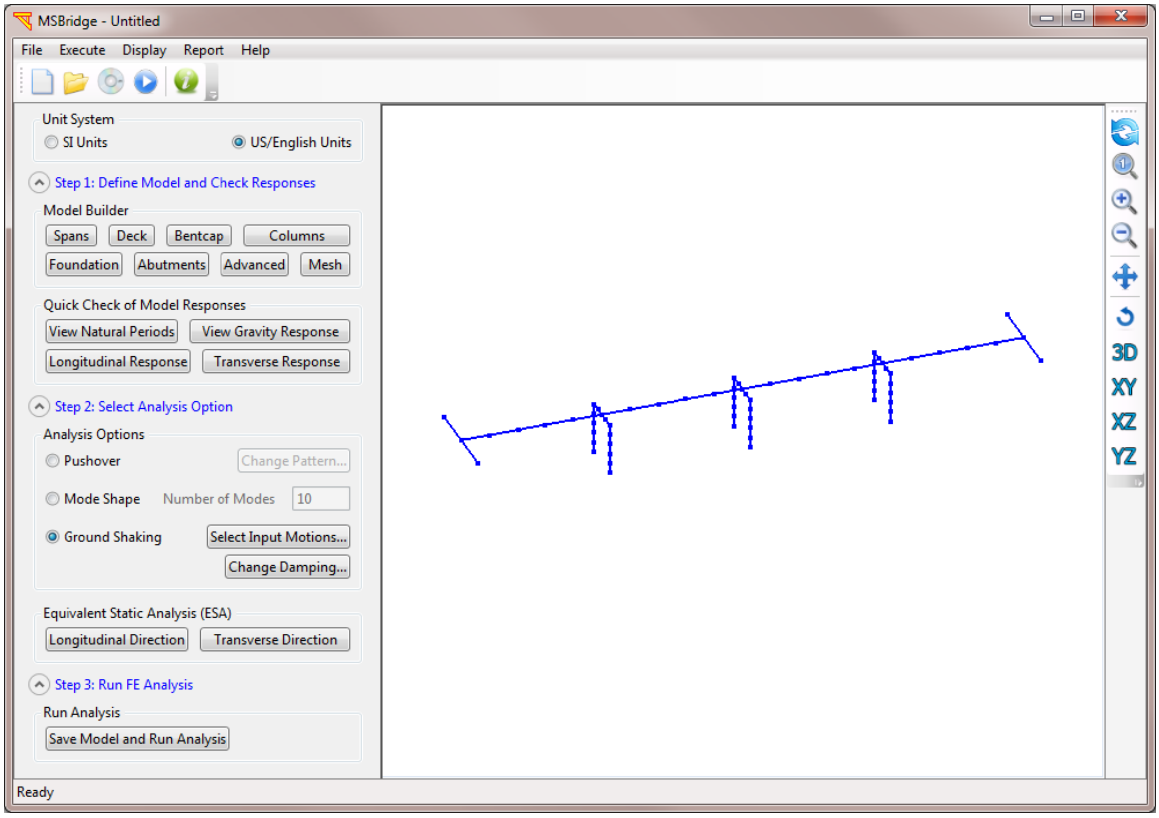


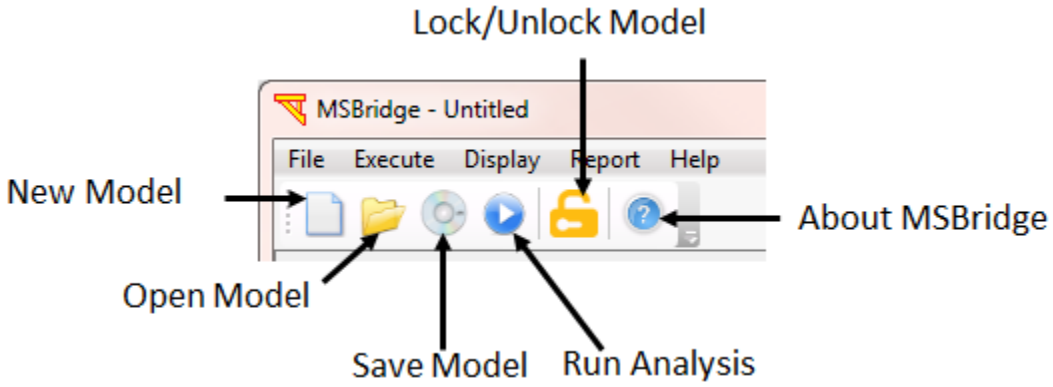
Fig. 2. MSBridge main window

2.2 Interface

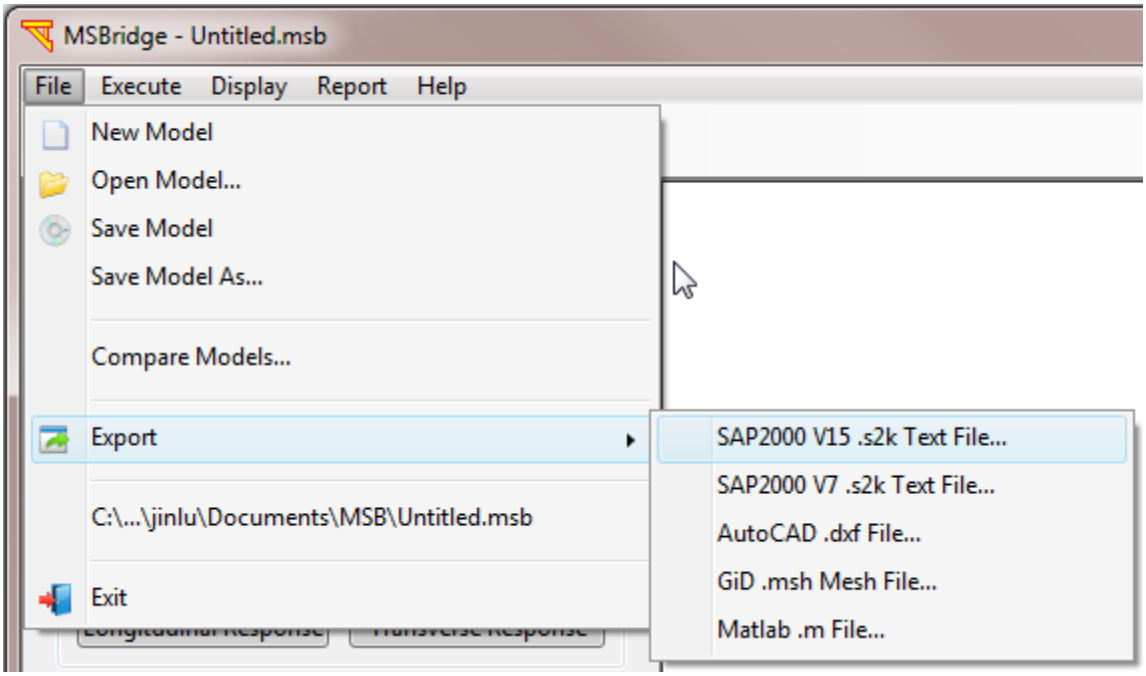
There are 3 main regions in the **MSBridge** window – menu bar, the model input, and the FE mesh.

2.2.1 Menu Bar

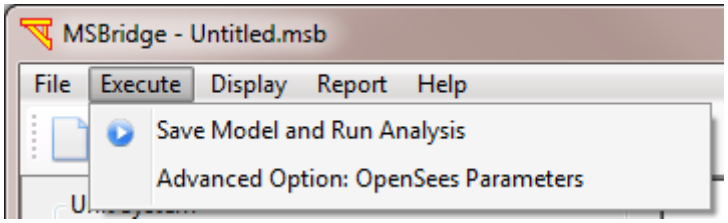
The menu bar, shown in Fig. 3, offers rapid access to most **MSBridge** main features.



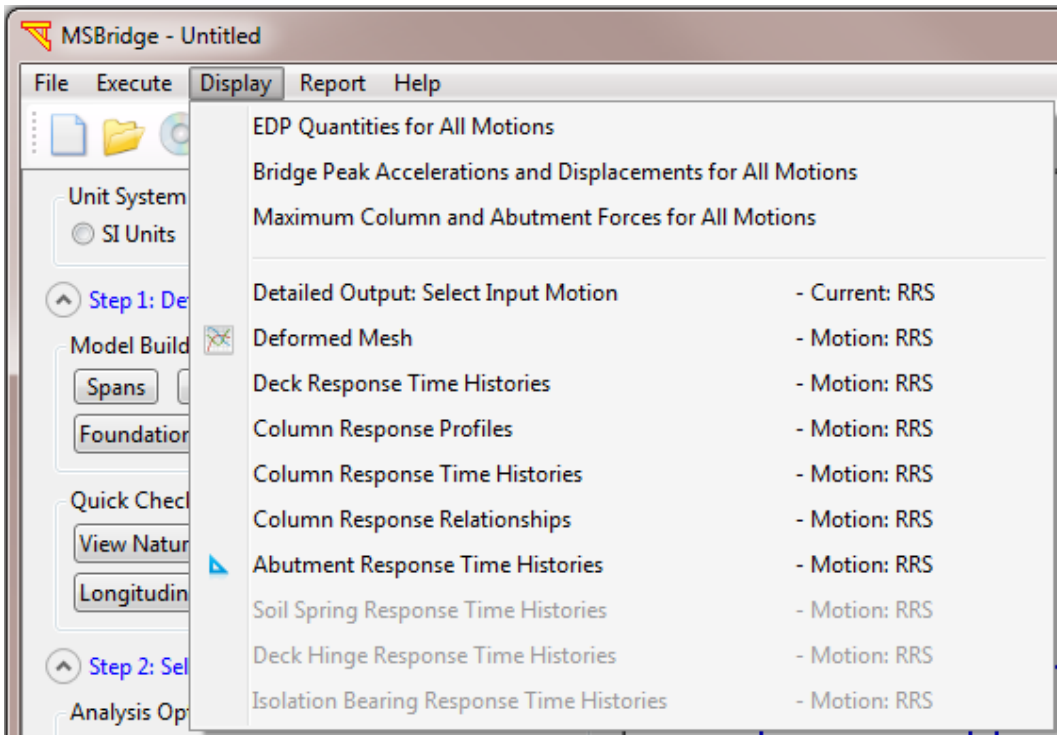
a)



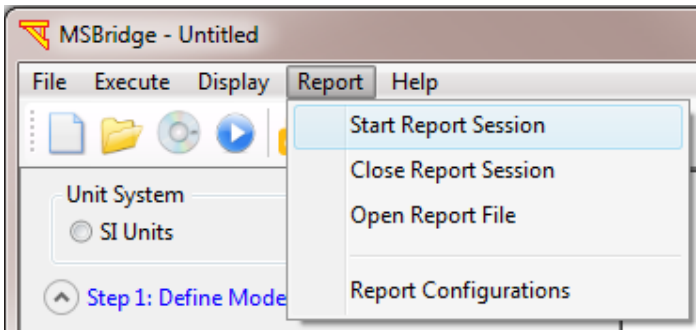
b)



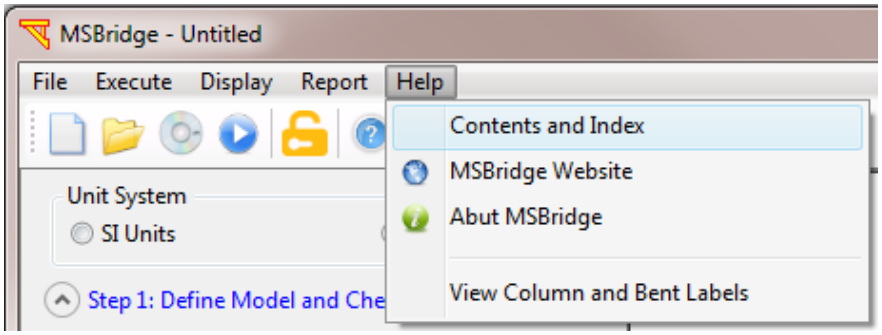
c)


MSBridge - Untitled
File
Execute
Display
Report
Help
EDP Quantities for All Motions
Bridge Peak Accelerations and Displacements for All Motions
Maximum Column and Abutment Forces for All Motions
Detailed Output: Select Input Motion - Current: RRS
Deformed Mesh - Motion: RRS
Deck Response Time Histories - Motion: RRS
Column Response Profiles - Motion: RRS
Column Response Time Histories - Motion: RRS
Column Response Relationships - Motion: RRS
Abutment Response Time Histories - Motion: RRS
Soil Spring Response Time Histories - Motion: RRS
Deck Hinge Response Time Histories - Motion: RRS
Isolation Bearing Response Time Histories - Motion: RRS
Unit System
SI Units
Step 1: De
Model Build
Spans
Foundatior
Quick Chec
View Natur
Longitudin
Step 2: Sel
Analysis Op

d)



e)



f)

Fig. 3. Menu and submenu bars: a) menu bar; b) menu **File**; c) menu **Execute**; d) menu **Display**; e) menu **Report**; and f) menu **Help**

The main features in **MSBridge** are organized into the following menus:

- **File:** Controls reading, writing and printing of model definition parameters, exporting the mesh to other software such as SAP2000 (for Versions 7 and 15) and Matlab, and exiting **MSBridge**.

Please note that exporting to SAP2000 .s2k file will work only if all of the following conditions are met (for now):

1) The column is linearly elastic
2) The abutment model is Elastic or Roller
3) The foundation must be Rigid-Base or Foundation Matrix
4) There is no Deck Hinge, no Isolation Bearing or no Steel Jacket, and
5) Analysis option is Pushover (monotonic) or Mode Shape Analysis

- **Execute:** Controls running analyses and OpenSees analysis parameters.
- **Display:** Controls displaying of the analysis results.
- **Report:** Controls creating the analysis report in Microsoft Word format
- **Help:** Visit the **MSBridge** website and display the copyright/acknowledgment message (Fig. 4).

Note that Fig. 3a shows a "Lock Model" button which is a toggle button that prevents from overwriting analysis results after the analysis is done. If the model is in "Locked Mode", all OK buttons (and Apply buttons) are disabled and users cannot make changes to the current model. To unlock the model, users need to click the "Lock Model" button. If the model is in "Unlocked Mode", analysis results (if any) will be overwritten if analysis is launched.

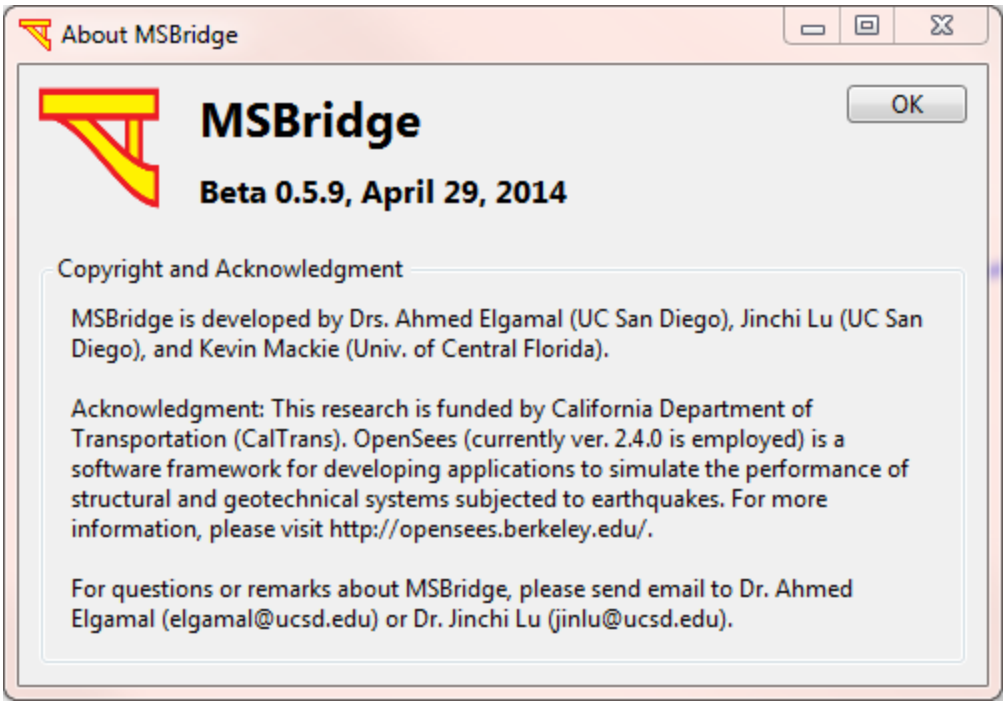

About MSBridge
MSBridge
Beta 0.5.9, April 29, 2014
OK
Copyright and Acknowledgment
MSBridge is developed by Drs. Ahmed Elgamal (UC San Diego), Jinchi Lu (UC San Diego), and Kevin Mackie (Univ. of Central Florida).
Acknowledgment: This research is funded by California Department of Transportation (CalTrans). OpenSees (currently ver. 2.4.0 is employed) is a software framework for developing applications to simulate the performance of structural and geotechnical systems subjected to earthquakes. For more information, please visit http://opensees.berkeley.edu/.
For questions or remarks about MSBridge, please send email to Dr. Ahmed Elgamal (elgamal@ucsd.edu) or Dr. Jinchi Lu (jinlu@ucsd.edu).

Fig. 4. MSBridge copyright and acknowledgment window

2.2.2 Model Input Region

The model input region controls definitions of the model and analysis options, which are organized into three regions (Fig. 2):

Step 1: Define Model & Check Responses: Controls definitions of bridge parameters including material properties. Meshing parameters are also defined in this step.
Step 2: Select Analysis Option: Controls analysis types (pushover analysis, mode shape analysis or ground shaking). Equivalent Static Analysis (ESA) option is also available.
Step 3: Run FE Analysis: Controls execution of the finite element analysis and display the analysis progress bar.

2.2.3 Finite Element Mesh Region

The Finite Element (FE) mesh region (Fig. 2) displays the generated mesh. In this window**,** the mesh can be manipulated by clicking buttons shown in Fig. 5.

The FE mesh shown in **MSBridge** is automatically generated. The user can also click the button at the top-right corner (shown in Fig. 5) to re-draw the FE mesh (based on the input data entered).

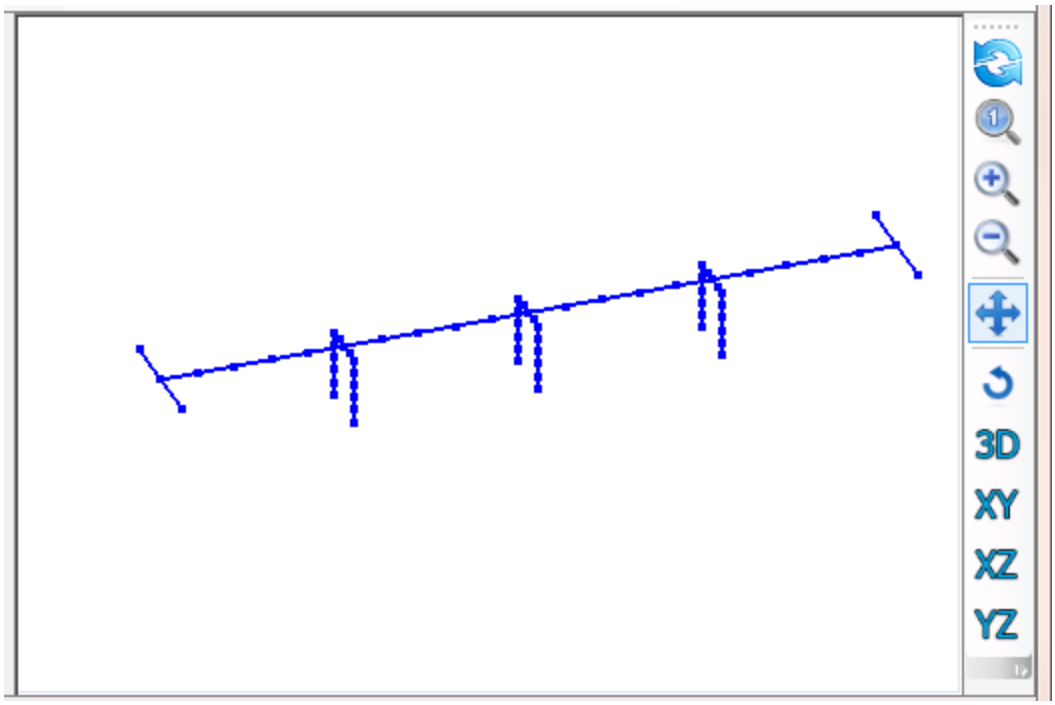


Fig. 5. Available actions in the FE Mesh window

3 Bridge Model

In **MSBridge**, the bridge deck, columns and bentcaps are modeled using beam-column elements. The foundation is fixed-based type by default (Fig. 2). Other available foundation types including soil springs and foundation matrix are modeled using zeroLength elements.

To define a bridge model, click corresponding buttons Fig. 6. To include a deck hinge, isolation bearing or use a non-zero skew angle for any bent or abutment, click **Advanced**. To change the numbers of beam-column element used for the deck, bentcaps and columns, click **Mesh**. Fig. 7 shows a bridge model with soil springs and deck hinges included.

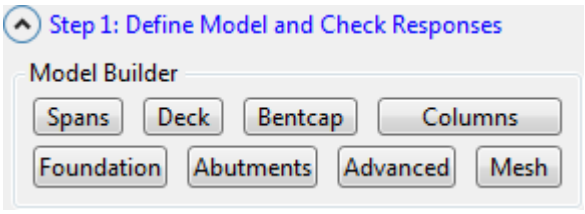


Fig. 6. Model builder buttons

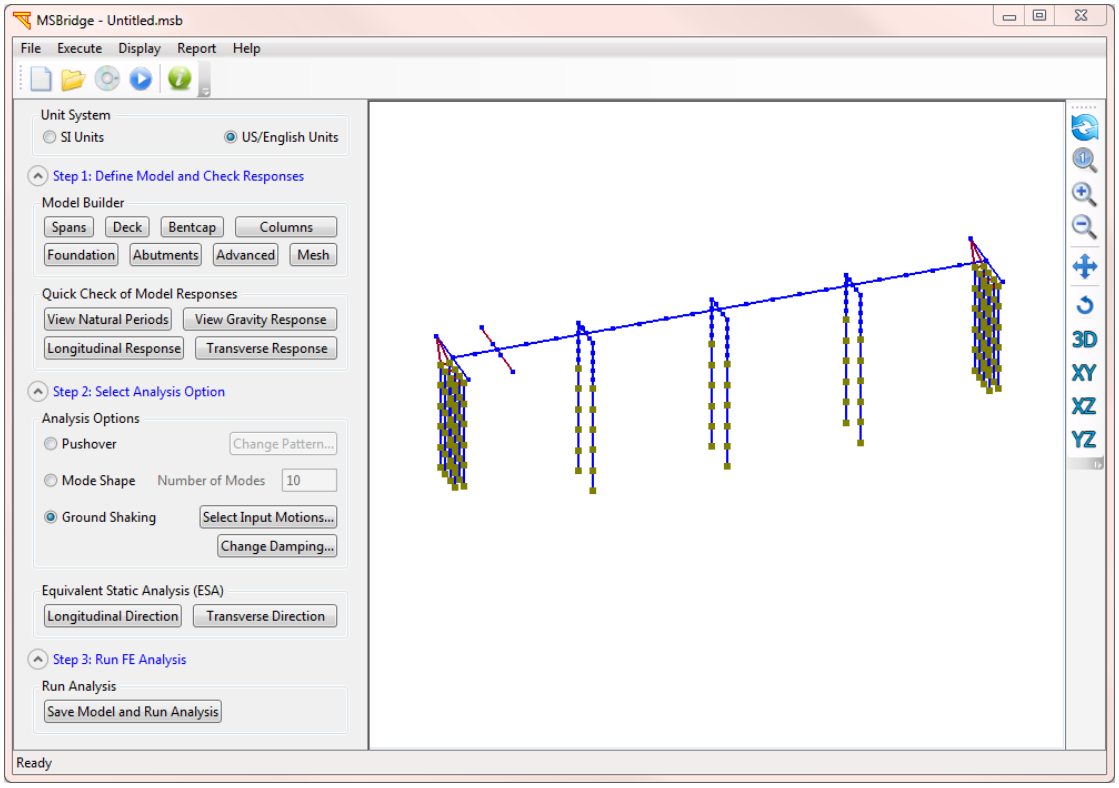


Fig. 7. MSBridge main window (bridge model with soil springs and deck hinges included)

3.1 Spans

To change the number of spans, click **Spans** in the main window (Fig. 6 and Fig. 8).

> **Number of Spans**: The total number of spans for a multi-span bridge. The minimum is 2 and the default value is 4. The maximum allowable number of spans is 100.

MSBridge supports models for both Straight Bridge and Curved Bridge options.

3.1.1 Straight Bridge

If the bridge has equal span lengths, click **Equal Span Length** and specify the span length (Fig. 8). The default is 60 feet.

If the bridge has varied span lengths, click **Varied Span Length** and then **Modify Span Lengths** to specify span lengths (Fig. 9). Fig. 10 shows a sample straight bridge model with varied span lengths.

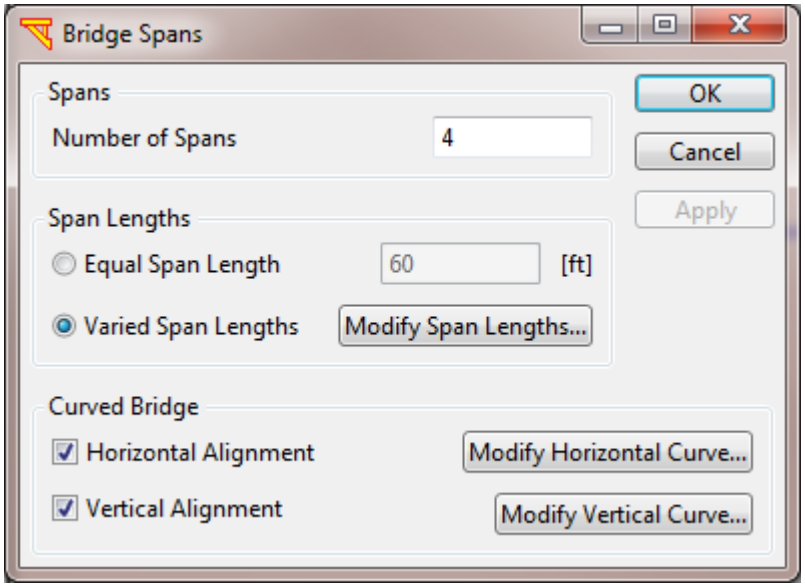


Fig. 8. Spans

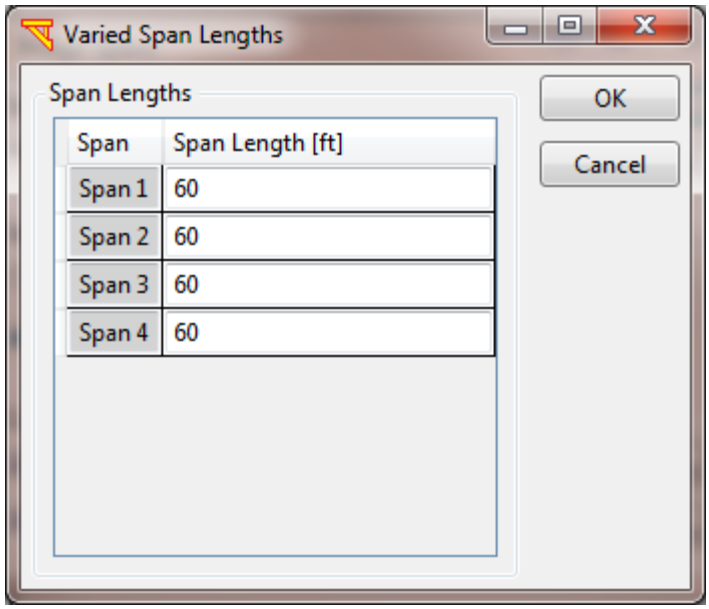


Fig. 9. Varied span lengths

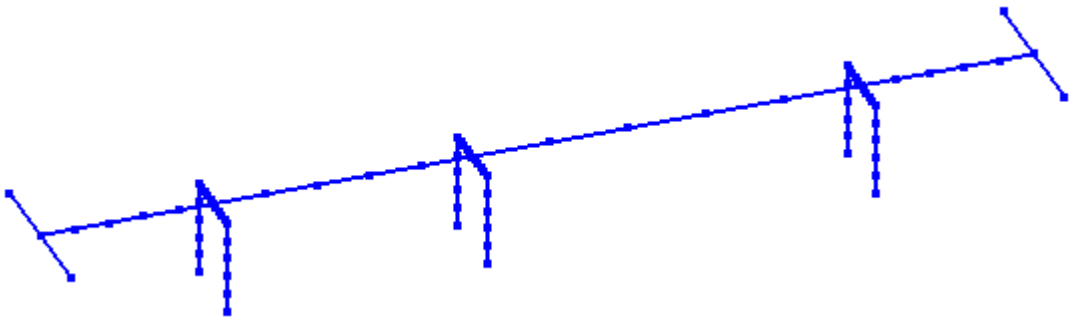


Fig. 10. Straight bridge with different span lengths

3.1.2 Curved Bridge

To define a curved bridge, please check **Horizontal Alignment** and/or **Vertical Alignment** in Fig. 8.

3.1.2.1 Horizontal Curved Bridge

To define a horizontally curved bridge, check **Horizontal Alignment** in Fig. 8. Fig. 11 shows the window to define the horizontal curves. **Begin Curve Length** refers to the starting location of the horizontal curve (see Fig. 12a). **Curve Radius** refers to the radius of the horizontal curve, **Curve Length** refers to the arc length of the horizontal curve. And the directions (**Left** or **Right**) refers to the arc rotation direction relative to the

starting location (**Right**: clockwise rotation in XY plan view; **Left**: counter-clockwise in XY plan view). Click **Insert Curve** to add a horizontal curve and click **Delete Curve** to remove the chosen curve. Fig. 14 shows examples of horizontal alignment.

3.1.2.2 Vertical Curved Bridge

To define a vertically curved bridge, check **Vertical Alignment** in Fig. 8. Fig. 13 shows the window to define the vertical curves. **Begin Curve Length** refers to the starting location of the beginning slope of the vertical curve (see Fig. 12b). **Curve Length** refers to the length of the vertical curve. **End Curve Slope** refers to the slope of the end slope. Note that the slope value can be negative, zero or positive. Similarly, Click **Insert Curve** to add a horizontal curve and click **Delete Curve** to remove the chosen curve. Fig. 15 shows examples of horizontal and vertical alignment.

Note that the horizontal curve/alignment employs the circular arc while the vertical curve/alignment employs the parabolic equation. Any two (horizontal or vertical) curves cannot be overlapped and any newly added curves must be located outside all previous curves.

For the detailed technical information on the horizontal & vertical alignments, please refer to the CalTrans Course Workbook for Land Surveyors (2011).

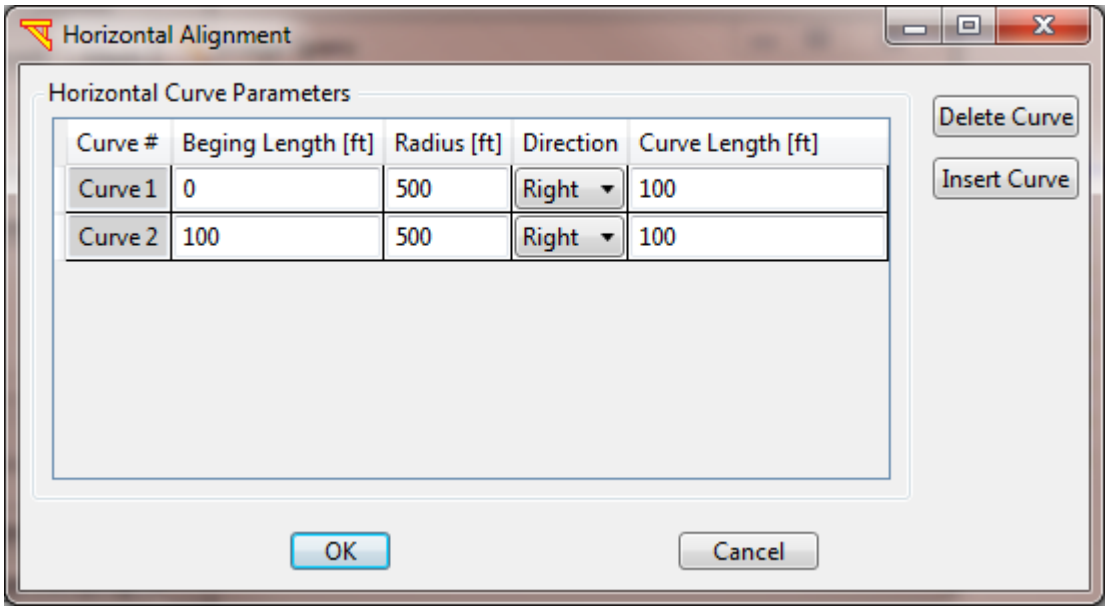
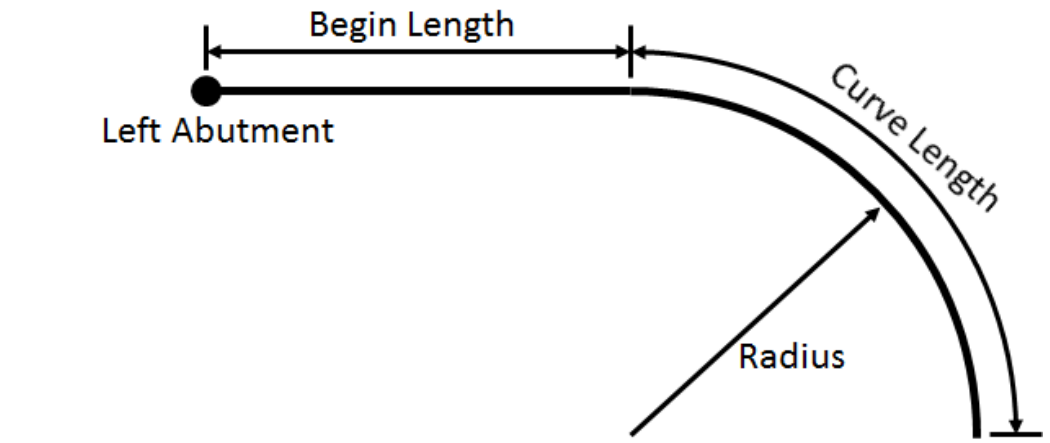
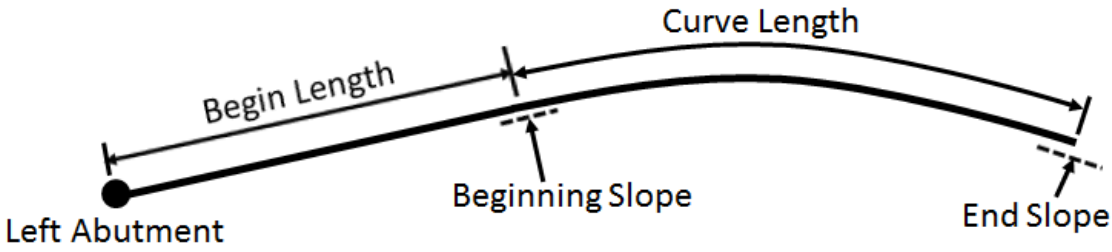


Fig. 11. Horizontal alignment



a)



b)

Fig. 12. Horizontal and vertical alignments: a) horizontal alignment (plan view); b) vertical alignment (side view)

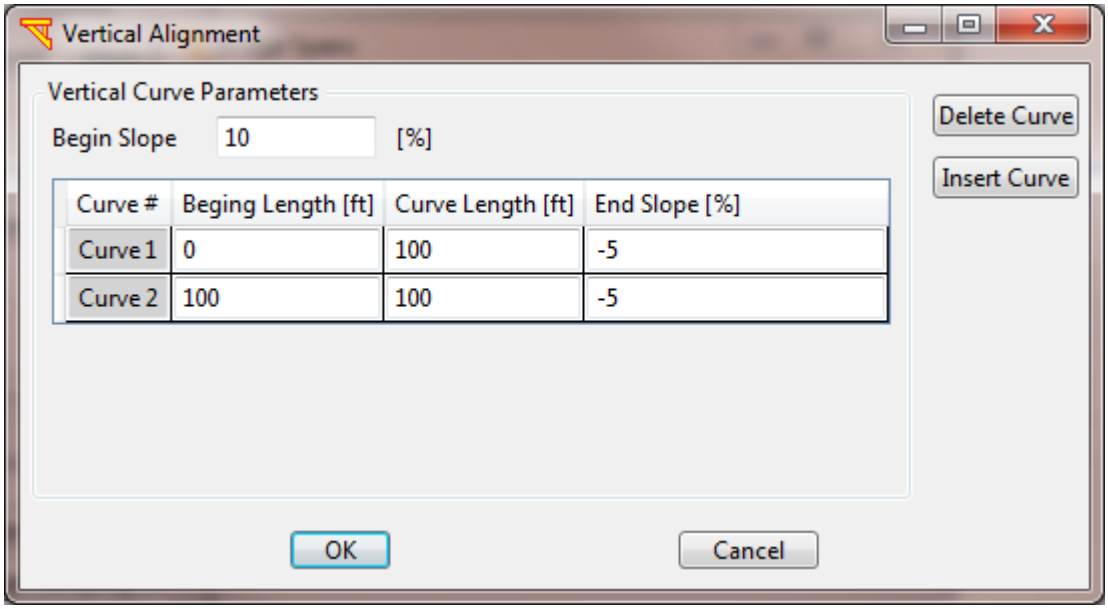
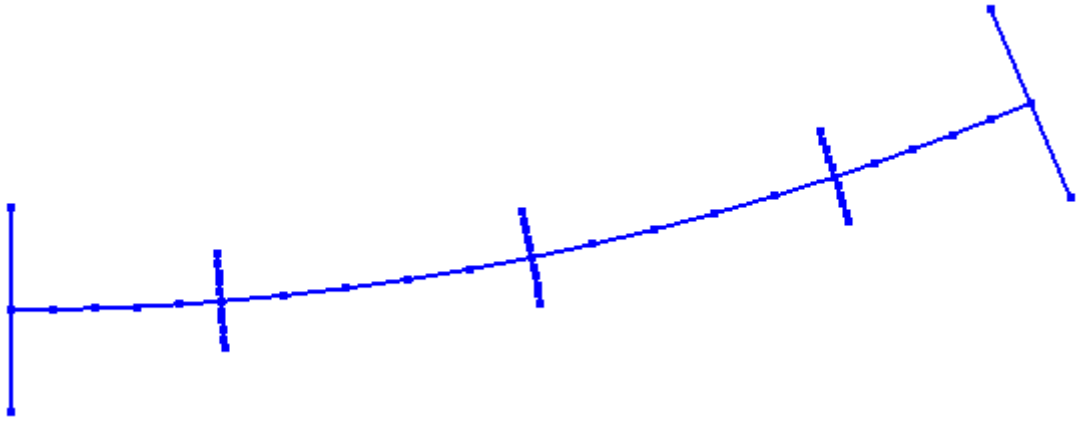
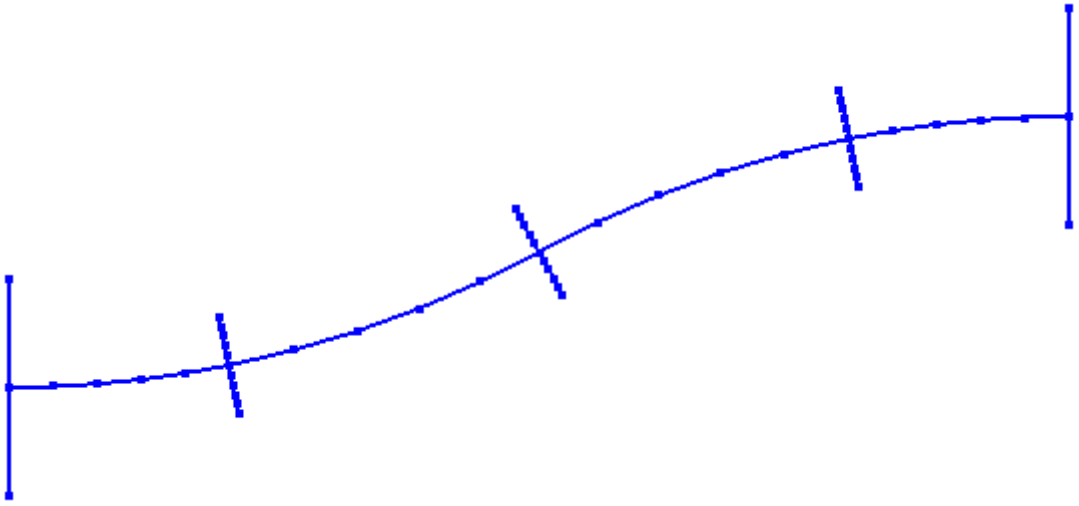


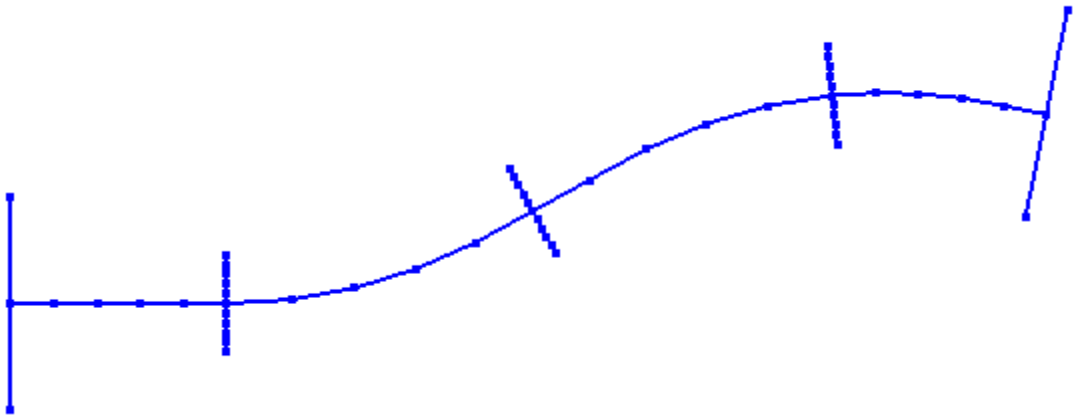
Fig. 13. Vertical alignment



a)



b)



c)

Fig. 14. Examples of horizontal curved bridges (horizontal alignment): a) single radius horizontal curve; b) multi-radius horizontal curve; c) horizontal curve connecting to straight parts).

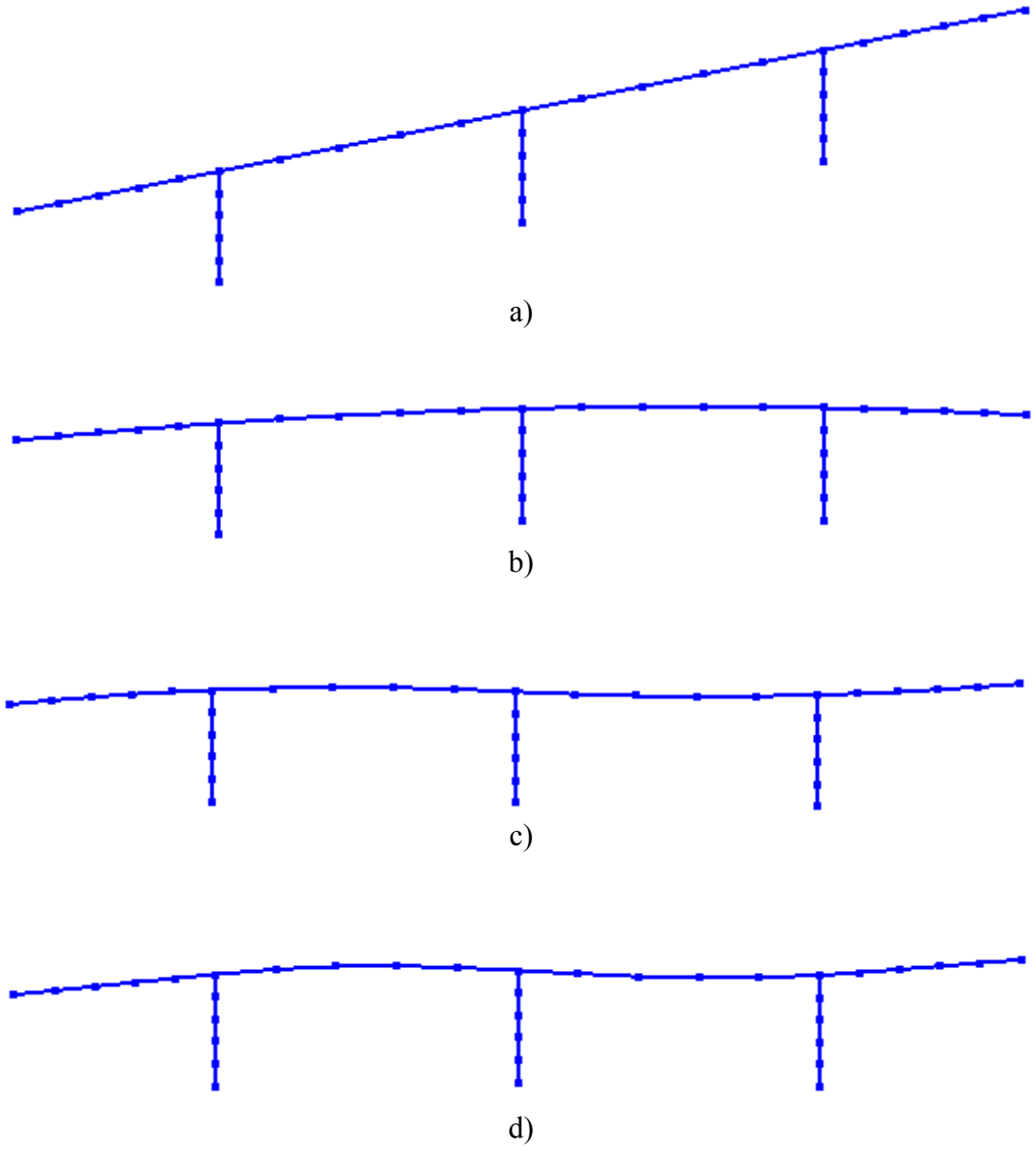


Fig. 15. Examples of vertically curved bridges (vertical alignment): a) single slope; b) begin and end slopes; c) multiple slopes; d) mixing slope and zero-slope.

3.2 Deck

To change Deck properties, click **Deck** in Fig. 6. Fig. 16 shows the window to modify the deck material and section properties.

MSBridge uses an elastic material model for the bridge deck elements. Fig. 16 shows the default values for the deck material properties including **Youngs Modulus**, **Shear Modulus**, and **Unit Weight**.

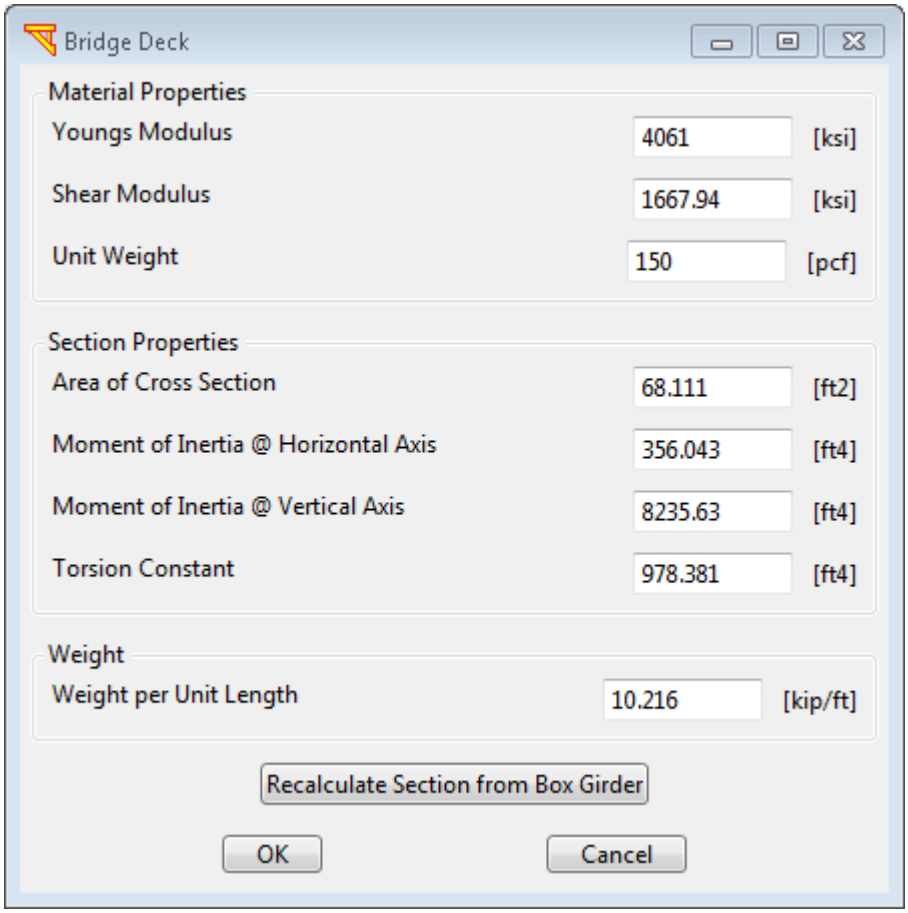


Fig. 16. Material properties of the bridge deck

Fig. 16 also shows the deck Section properties. Section properties can be input directly in Fig. 16, if available. If this information is not available **MSBridge** will generate properties based on general box girder section dimensions. Click **Recalculate Section from Box Girder** in Fig. 16 to define the new box girder shape (Fig. 17). The default values of geometrical properties are of typical for a four-cell reinforced concrete box girder deck configuration. **Weight per Unit Length** is equal to the **Area of Cross Section** times the **Unit Weight** defined in Fig. 16. Click **OK** in Fig. 17 if the user would like to use the defined cross section. Corresponding entries in Fig. 16 will be updated.

IMPORTANT NOTE: If using self-defined section properties the **Box Width** in Fig. 17 will be used as the deck width of the bridge. To use a different deck width, the user needs to modify **Box Width** in Fig. 17.

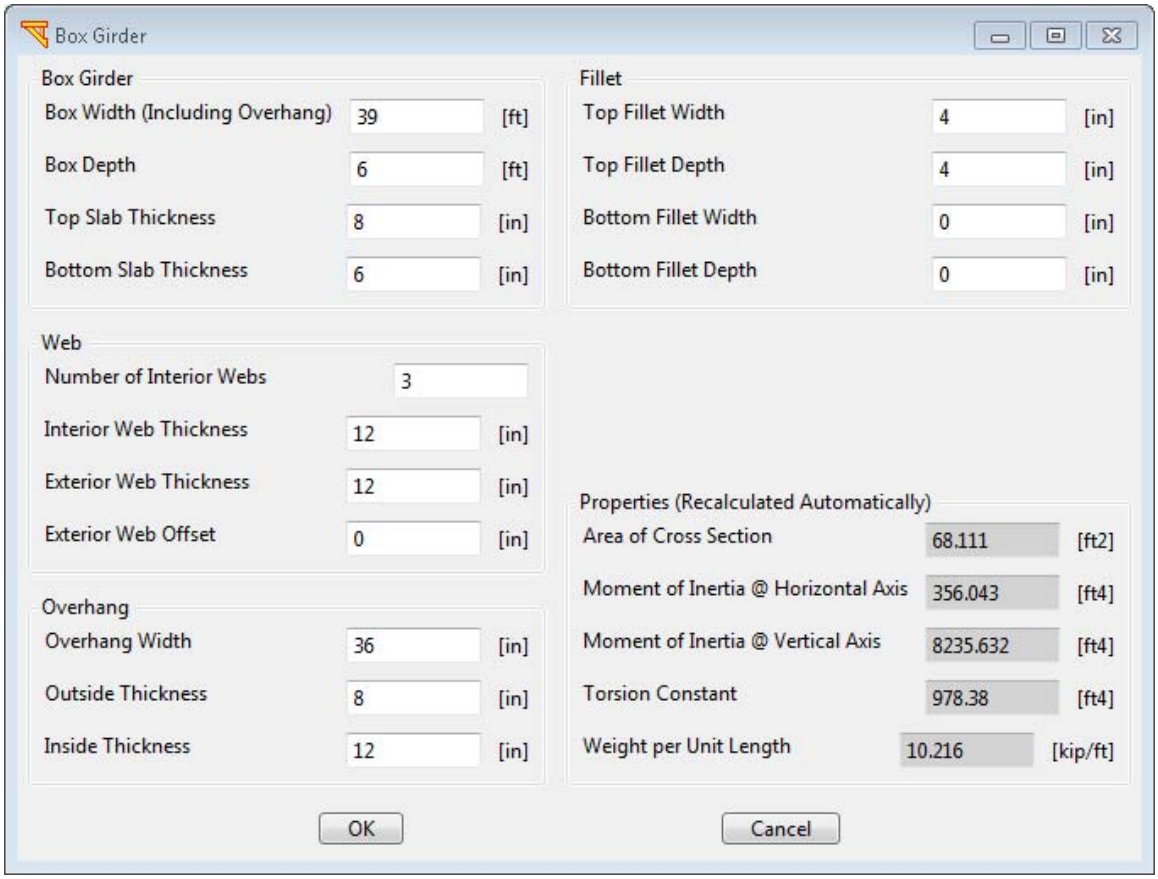

Box Girder
Box Girder
Box Width (Including Overhang) 39 [ft]
Box Depth 6 [ft]
Top Slab Thickness 8 [in]
Bottom Slab Thickness 6 [in]
Fillet
Top Fillet Width 4 [in]
Top Fillet Depth 4 [in]
Bottom Fillet Width 0 [in]
Bottom Fillet Depth 0 [in]
Web
Number of Interior Webs 3
Interior Web Thickness 12 [in]
Exterior Web Thickness 12 [in]
Exterior Web Offset 0 [in]
Overhang
Overhang Width 36 [in]
Outside Thickness 8 [in]
Inside Thickness 12 [in]
Properties (Recalculated Automatically)
Area of Cross Section 68.111 [ft2]
Moment of Inertia @ Horizontal Axis 356.043 [ft4]
Moment of Inertia @ Vertical Axis 8235.632 [ft4]
Torsion Constant 978.38 [ft4]
Weight per Unit Length 10.216 [kip/ft]
OK
Cancel

Fig. 17. Box girder shape employed for the bridge deck

3.3 Bentcap

To change bent cap properties, click **Bentcap** in Fig. 6. Fig. 18 shows the window to modify the bentcap material and section properties.

MSBridge uses an elastic material model for the bridge bentcap elements. Fig. 18 shows the default values for the bentcap material properties including **Youngs Modulus**, **Shear Modulus**, and **Unit Weight**.

Fig. 18 also shows the bentcap Section properties. Section properties can be input directly in Fig. 18, if available. If this information is not available **MSBridge** will generate properties based on a rectangular section dimensions. Click **Recalculate Section from Rectangular** in Fig. 18 to define the new rectangular shape (Fig. 19). **Weight per Unit Length** is equal to the **Area of Cross Section** times the **Unit Weight** defined in Fig. 18.

Click **OK** in Fig. 17 if the user would like to use the defined cross section. Corresponding entries in Fig. 18 will be updated.

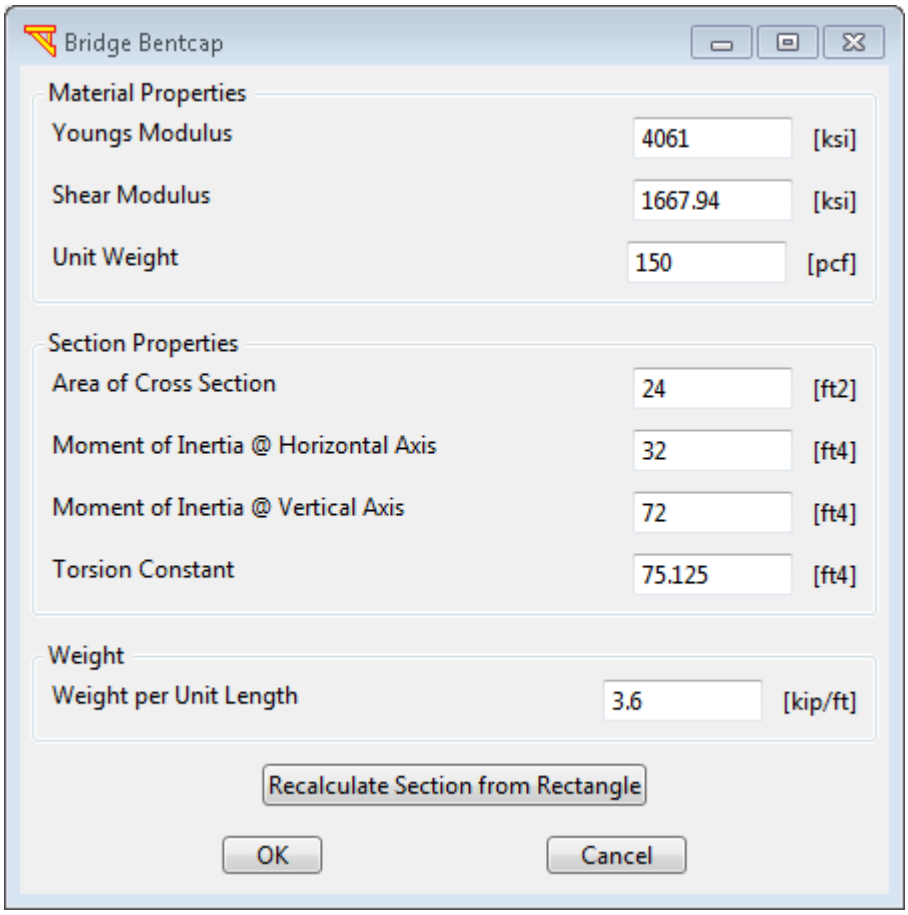


Fig. 18. Material properties of the bent cap

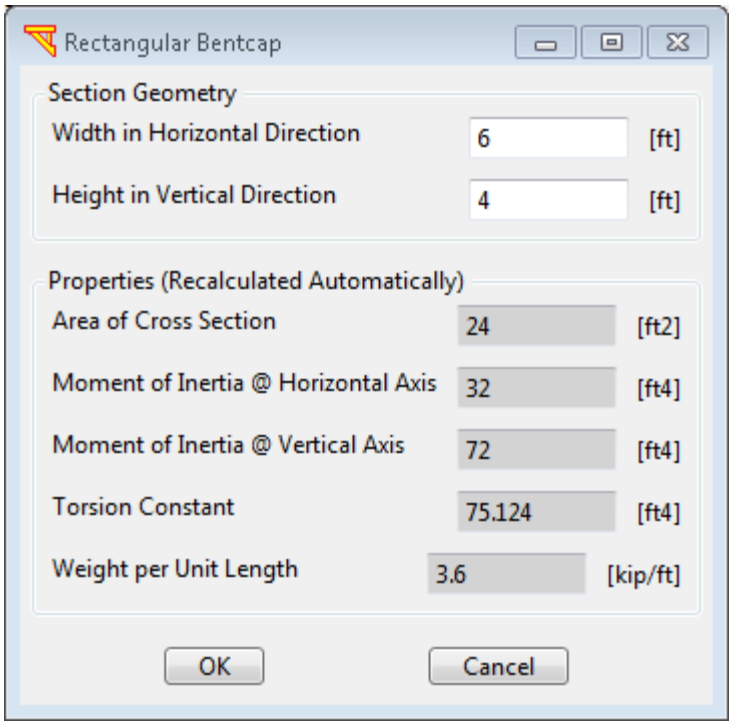


Fig. 19. Rectangular shape employed for the bent cap

3.4 Columns

To modify column properties, click **Columns** in Fig. 6. Fig. 20 shows the window to define columns.

The current version assumes that all bents have the same number of columns, and the same **Column Spacing**. If **Number of Column for Each Bent** is 1, **Column Spacing** will be ignored (Fig. 20).

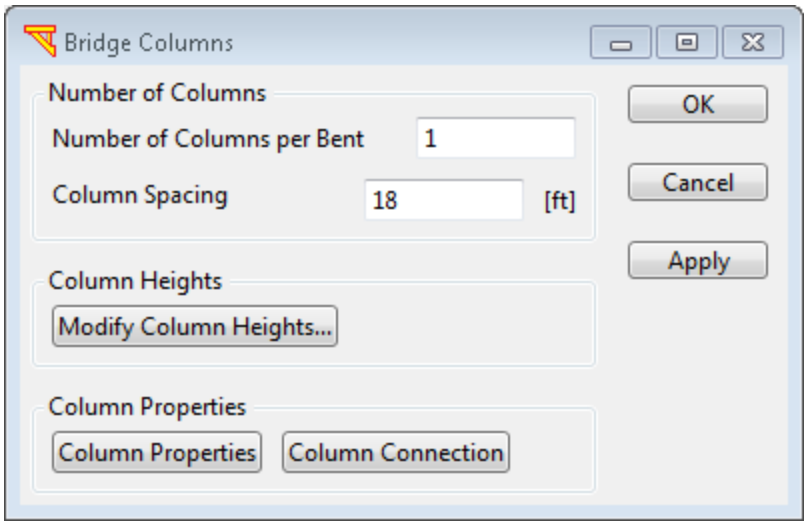


Fig. 20. Columns

3.4.1 Column Heights

To define column heights, click **Modify Column Heights** in Fig. 20. A window for defining column heights will appear (Fig. 21).

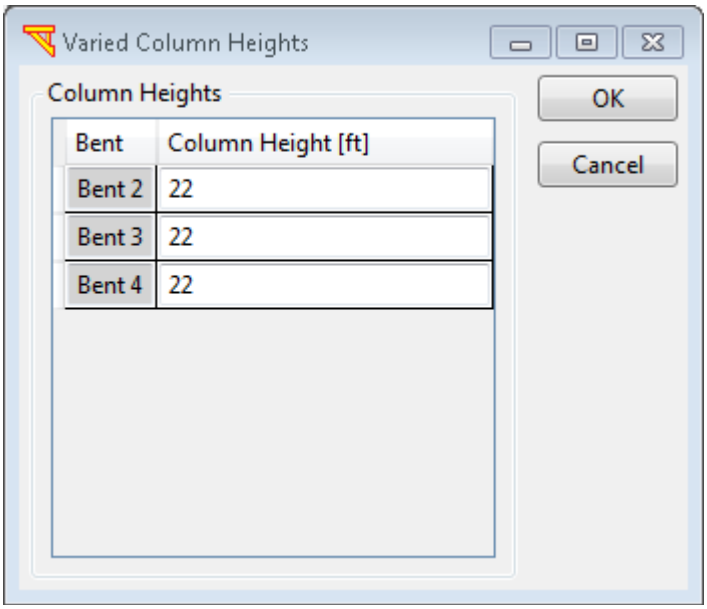


Fig. 21. Column heights

3.4.2 Column Connection

In a multi-column case (the number of columns per bent is equal to 2 or more), the user can specify the boundary connection conditions of the columns. Click **Column Connection** (in Fig. 20) to select the boundary condition for the columns and bent cap connection. Three options are available (Fig. 22): i) fixed at top / pinned at base; ii) pinned at top / fixed base; and iii) fixed at both top and base. **Note:** In a single column case (the number of columns per bent is equal to 1), both column top and base are assumed fixed.

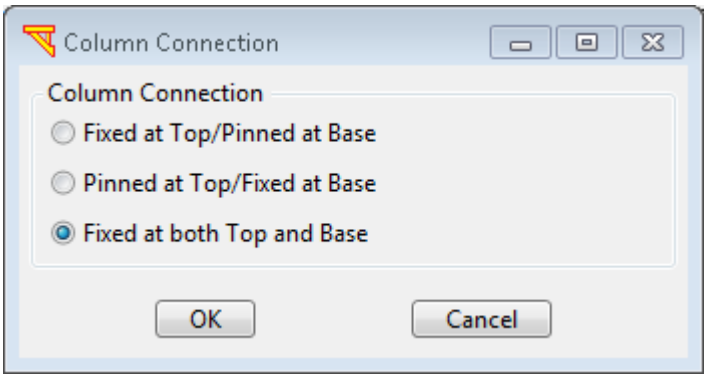


Fig. 22. Column boundary conditions

3.4.3 Column Properties

To define the material and geometrical properties of column, click **Column Properties** in Fig. 20. For now, all columns will assume to have the same material and geometrical properties. Uses can choose to use the linearly elastic column or nonlinear Fiber column. By default, the nonlinear Fiber section is used (Fig. 23).

3.4.3.1 Cross Section Types

The cross sections currently available in **MSBridge** include **Circle**, **Octagon**, **Hexagon** and **Rectangle** (Fig. 23). For the Circular, Octagonal and Hexagonal sections, the user needs to define the Column Diameter. For Rectangular section, the user needs to define the widths in bridge longitudinal and transverse directions (Fig. 23).

3.4.3.2 Linearly Elastic Column

To activate the linear column, check the checkbox **Column is Linearly Elastic (**Fig. 24). Elastic beam-column element (**elasticBeamColumn**, McKenna et al. 2010) is used for the column in this case.

Click **Elastic Material Properties** to define **Youngs Modulus, Shear Modulus** and **Unit Weight** of the column (Fig. 25). Click **Section Properties** to change the column section properties (by changing the cracked section factors) as shown in Fig. 26.

3.4.3.3 Nonlinear Fiber Section

To use nonlinear Fiber section for the column, click **Nonlinear Fiber Section** (Fig. 23). The window for defining the Fiber section is shown in Fig. 27. Click Material Properties buttons to define the material properties for the rebar, the core and the cover concrete (Fig. 28). Nonlinear beam-column elements with fiber section for the circular cross section (Fig. 29) are used to simulate the column in this case. The calculations of fibers for the octagonal and hexagonal cross sections are similar to that of the circular cross section except for the cover. Fig. 30 shows a slightly treatment of fiber calculations for the octagonal and hexagon cross sections.

For Rectangular section, the number of bars refers to the number of reinforcing bars around the section perimeter (equal spacing).

Two types of nonlinear Beam-Column Elements are available for the column: **Beam With Hinges** and **Force-Based Beam Column** (McKenna et al. 2010). By default, Forced-based beam-column elements (**nonlinearBeamColumn**, McKenna et al. 2010) are used (the number of integration points = 5). The default values for the material properties of the column are shown in Tables 2-4.

When the **Beam With Hinges** Element is used, the calculation of the plastic hinge length (L_p) for the column is based on Eq. 7.25 of SDC (2010):

$$L_p = \begin{cases} 0.08L + 0.15 f_{ye} d_{bl} \geq 0.3 f_{ye} d_{bl} & (\text{in}, \text{ksi}) \\ 0.08L + 0.022 f_{ye} d_{bl} \geq 0.044 f_{ye} d_{bl} & (\text{mm}, \text{MPa}) \end{cases}$$

Where L is the column height, f_{ye} is the steel yield strength, d_{bl} is the longitudinal bar size. The plastic hinge length (L_p) is the equivalent length of column over which the plastic curvature is assumed constant for estimating plastic rotation (SDC 2010).

The material options available for the steel bar include **Elastic**, **Steel01**, **Steel02** and **ReinforcingSteel**. The material options available for the concrete include **Elastic**, **ENT** (Elastic-No Tension), **Concrete01**, and **Concrete02**.

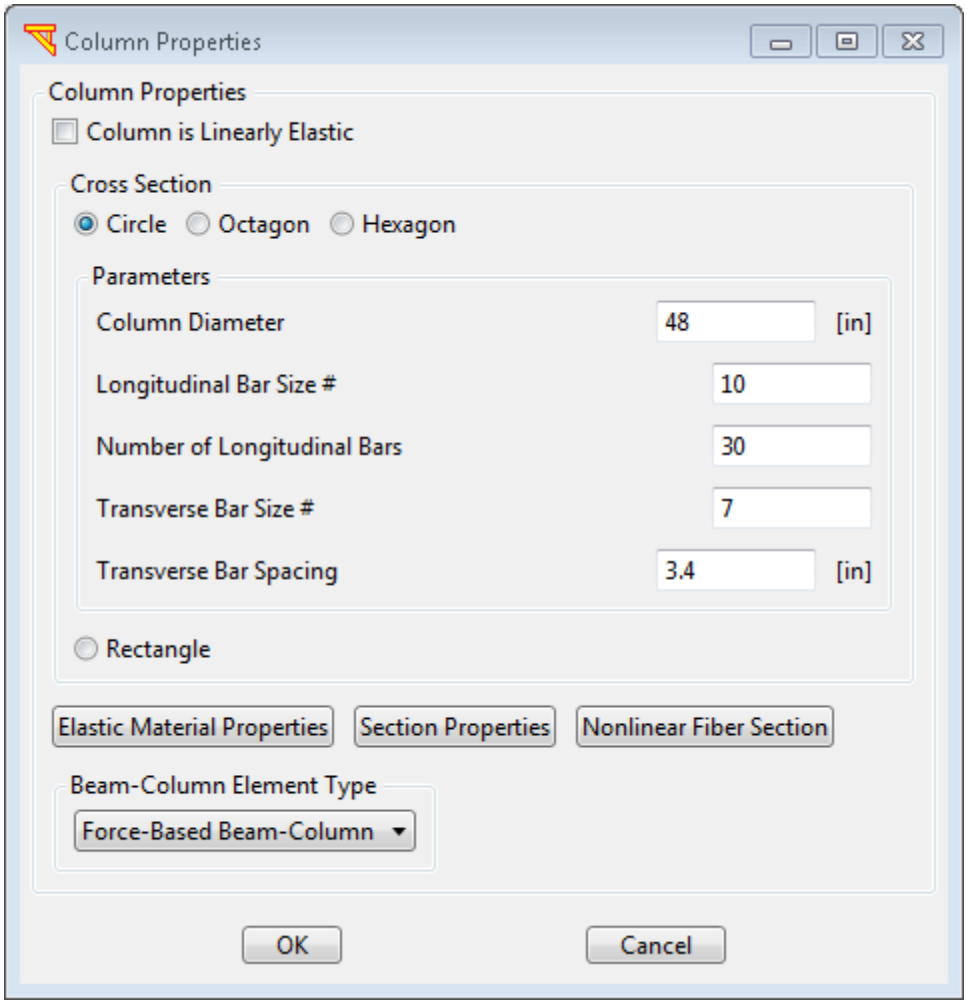


Fig. 23. Column properties and available beam-column element types

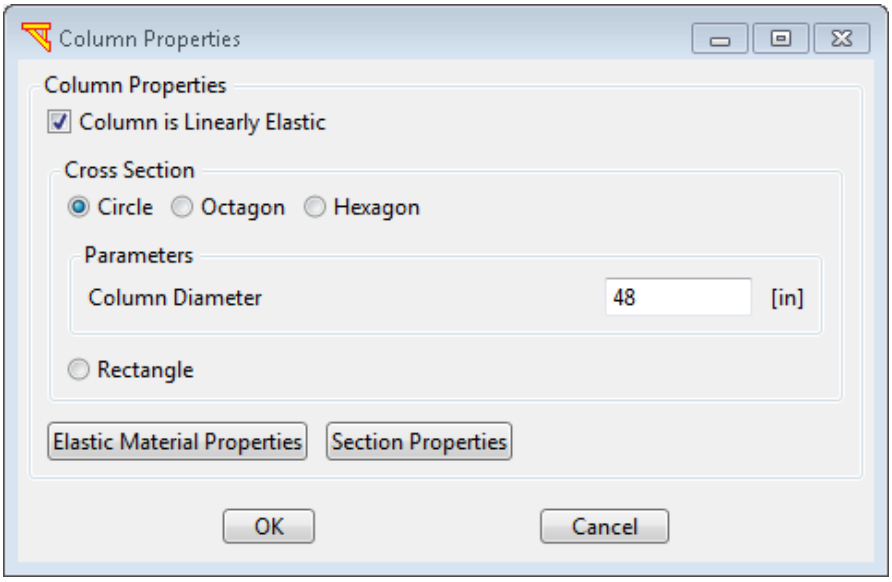


Fig. 24. Definition of linear column

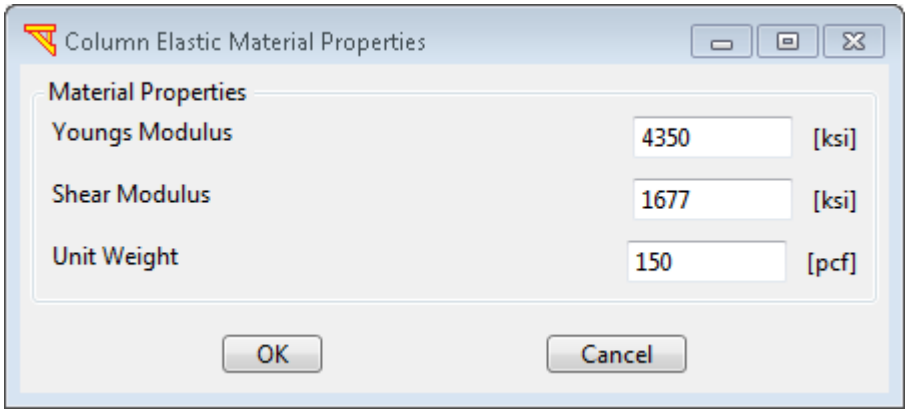


Fig. 25. Column Elastic material properties

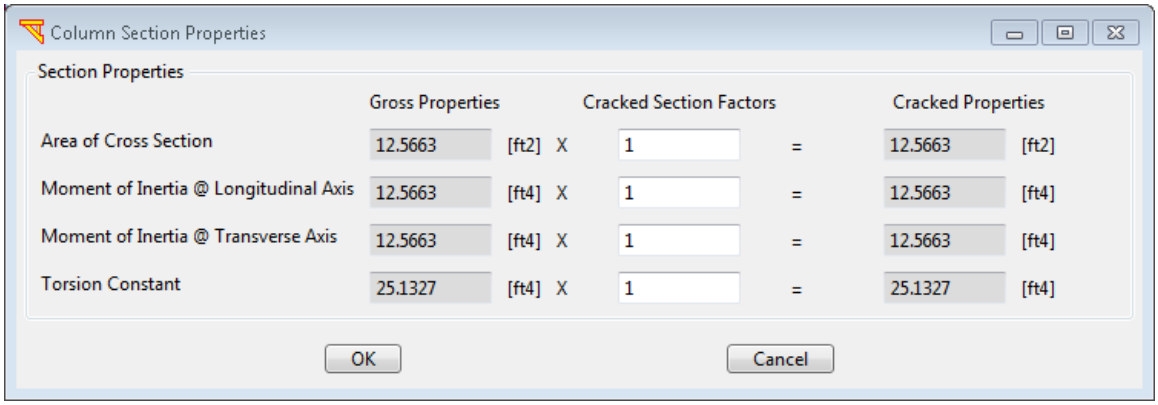


Fig. 26. Column Section properties

By default, the Steel02 material in OpenSees (McKenna et al. 2010) is employed to simulate the steel bars and Concrete02 material is used for the concrete (core and cover). Steel02 is a uniaxial Giuffré-Menegotto-Pinto material that allows for isotropic strain hardening. Concrete02 is a uniaxial material with linear tension softening. The Concrete02 material parameters were obtained from the Mander (1988) constitutive relationships for confined and unconfined concrete. More details on the derivation of the default values and the OpenSees uniaxialMaterial definitions used for each material are shown in Appendix A.

Fig. 31, Fig. 32, and Fig. 33 show the stress-strain curves for the steel, core, and cover concrete materials, respectively (The stress-strain curve is only calculated up to 6% of strain). These plots can be obtained for updated material properties directly from the interface by clicking on the corresponding **View Stress-Strain** buttons in the Column Material Properties window (Fig. 27). The moment-curvature response for the column is shown in Fig. 34 (generated with consideration of the overall deck weight 2680 kip applied at the column top). For comparison, XSECTION (CalTrans, 1999) result is also available (Fig. 34).

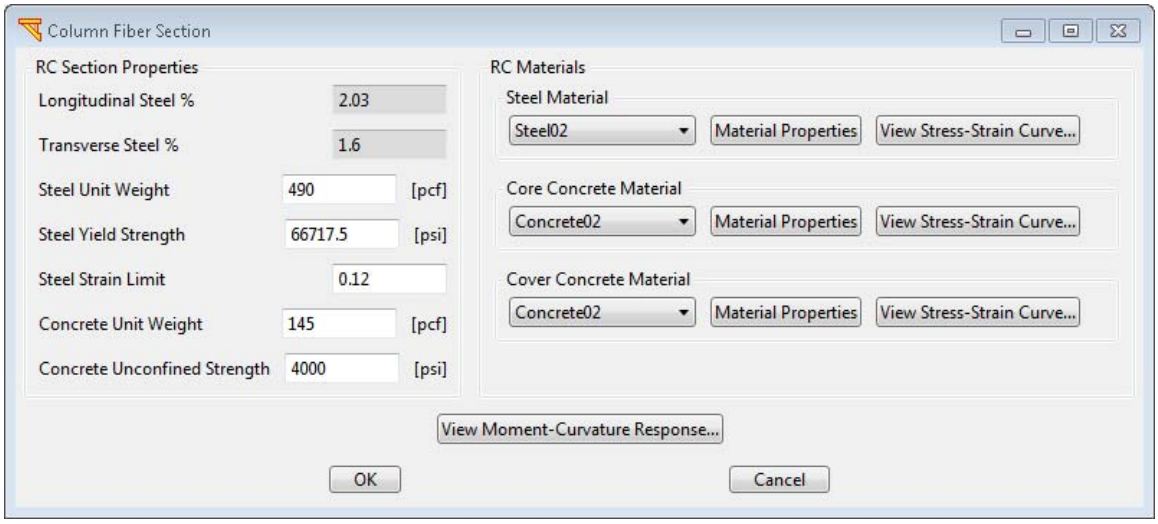
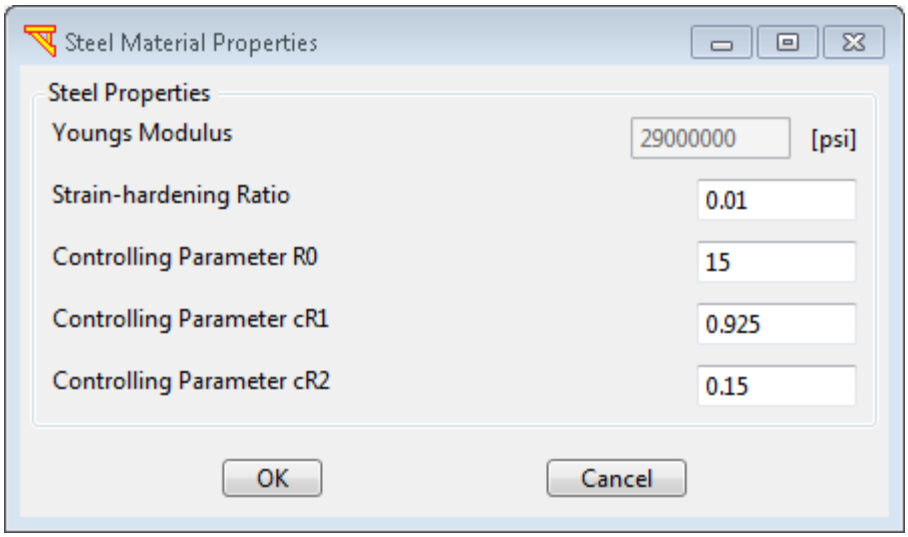
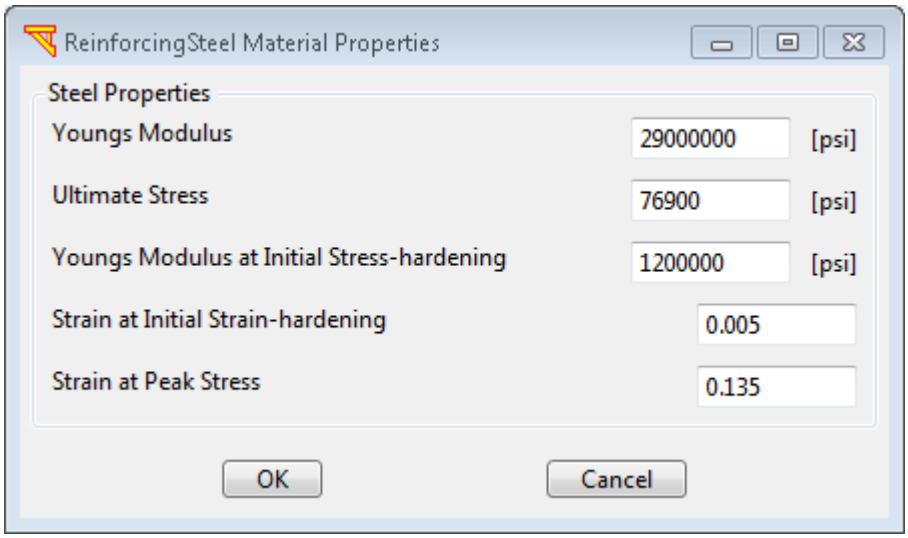


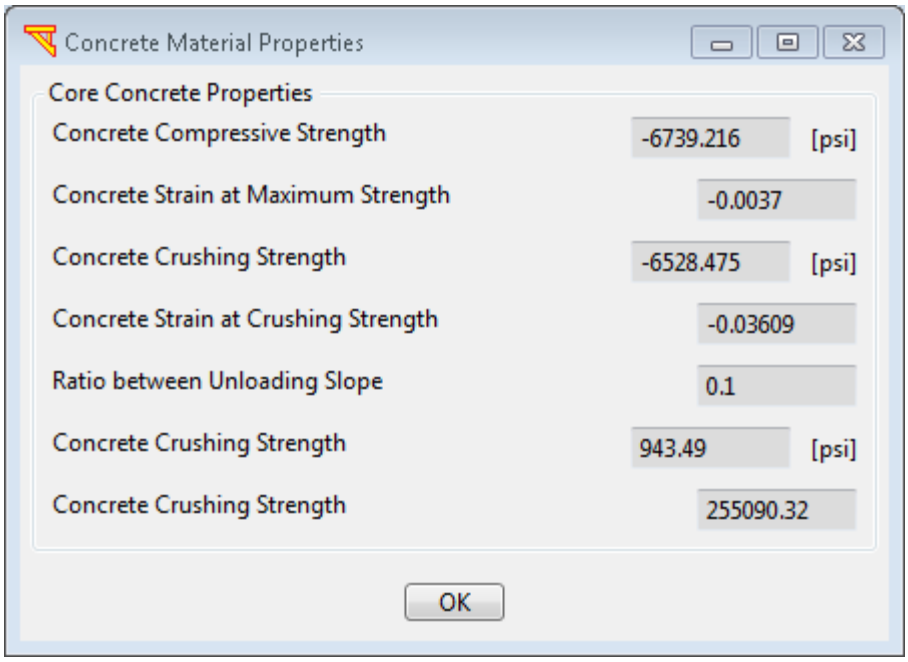
Fig. 27. Nonlinear Fiber Section window


Steel Material Properties
Steel Properties
Youngs Modulus
29000000
[psi]
Strain-hardening Ratio
0.01
Controlling Parameter R0
15
Controlling Parameter cR1
0.925
Controlling Parameter cR2
0.15
OK
Cancel

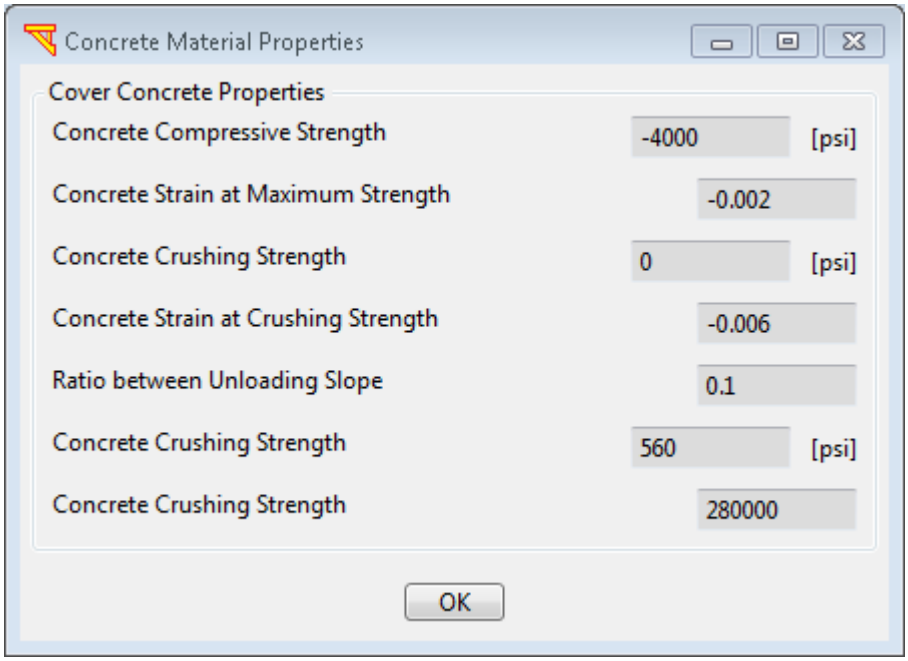
a)


ReinforcingSteel Material Properties
Steel Properties
Youngs Modulus
29000000
[psi]
Ultimate Stress
76900
[psi]
Youngs Modulus at Initial Stress-hardening
1200000
[psi]
Strain at Initial Strain-hardening
0.005
Strain at Peak Stress
0.135
OK
Cancel

b)



c)



d)

Fig. 28. Column nonlinear material properties: a) Steel02 material; b) ReinforcingSteel material; c) Concrete02 material for the core concrete; d) Concrete02 material for the cover material

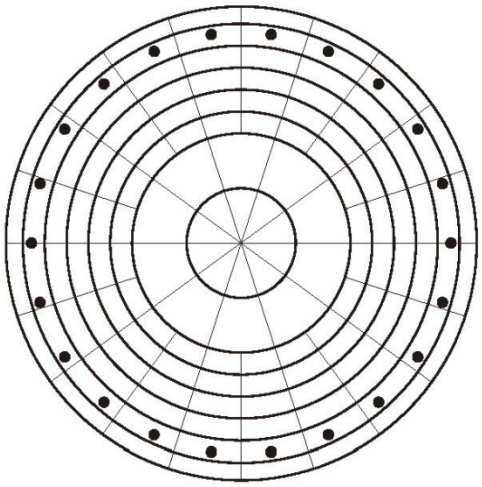
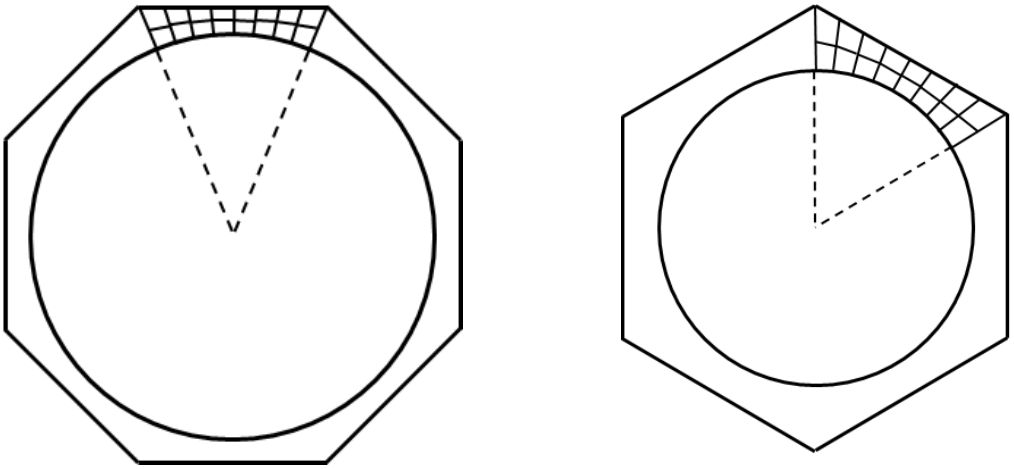


Fig. 29. Column fiber section (based on PEER best modeling practices report, Berry and Eberhard, 2007): a) Circle; b) Octagon; c) Hexagon



a) b)

Fig. 30. OpenSees *quadrilateral patch* employed for calculating the cover concrete fibers for: a) Octagon; and b) Hexagon cross section

Table 1. Default values for column reinforced concrete (RC) section properties

Parameter	Value
Longitudinal bar size (US #)	10
Longitudinal steel %	2
Transverse bar size (US #)	7
Transverse steel %	1.6
Steel unit weight (pcf)	490
Steel yield strength (psi)	66717.5
Concrete unit weight (pcf)	145
Concrete unconfined strength (psi)	4000

Table 2. Default values for Steel02 material properties

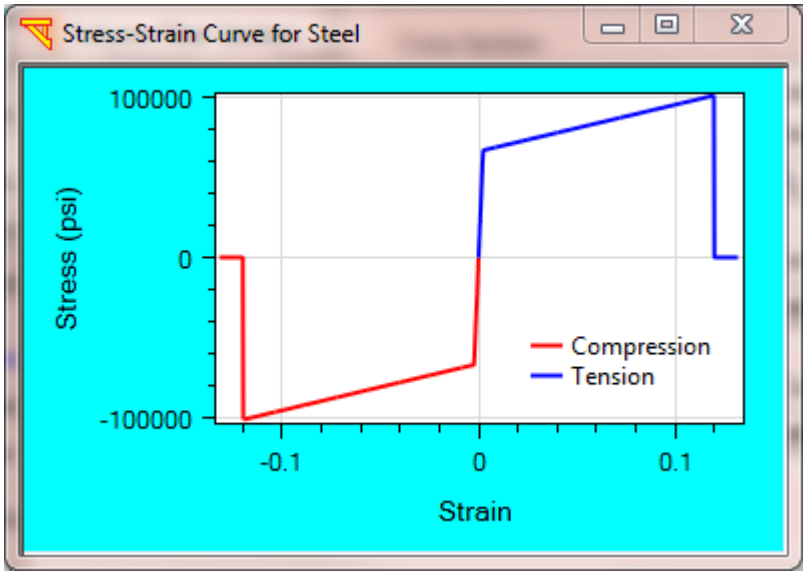
Parameter	Value	Typical range
Steel yield strength (psi)	66717.5	50,000-68,000
Young's modulus (psi)	29,000,000	-
Strain-hardening ratio*	0.01	0.005-0.025
Controlling parameter R0**	15	10-20
Controlling parameter cR1**	0.925	--
Controlling parameter cR2**	0.15	--

*The strain-hardening ratio is the ratio between the post-yield stiffness and the initial elastic stiffness.

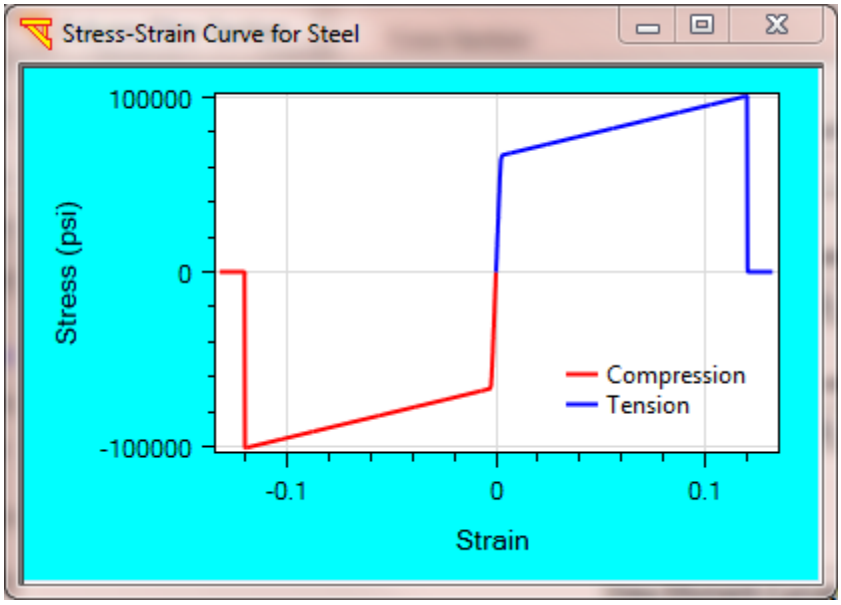
**The constants R0, cR1 and cR2 are parameters to control the transition from elastic to plastic branches.

Table 3. Default values for Concrete02 material properties

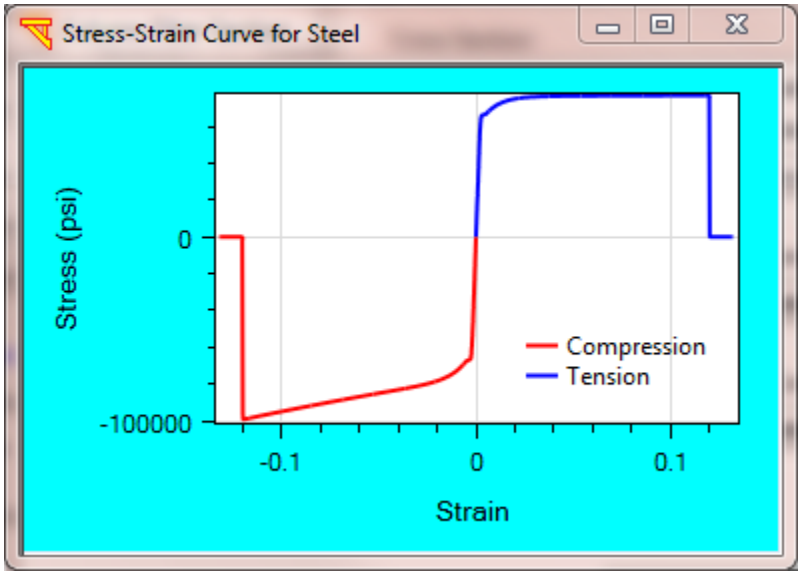
Parameter	Core	Cover
Elastic modulus (psi)	3,644,147	3,644,147
Compressive strength (psi)	-6,739	-4000
Strain at maximum strength	-0.0037	-0.002
Crushing strength (psi)	-6,538	0
Strain at crushing strength	-0.036	-0.006
Ratio between unloading slope	0.1	0.1
Tensile strength (psi)	943.49	560
Tensile softening stiffness (psi)	255,090	280,000



a)

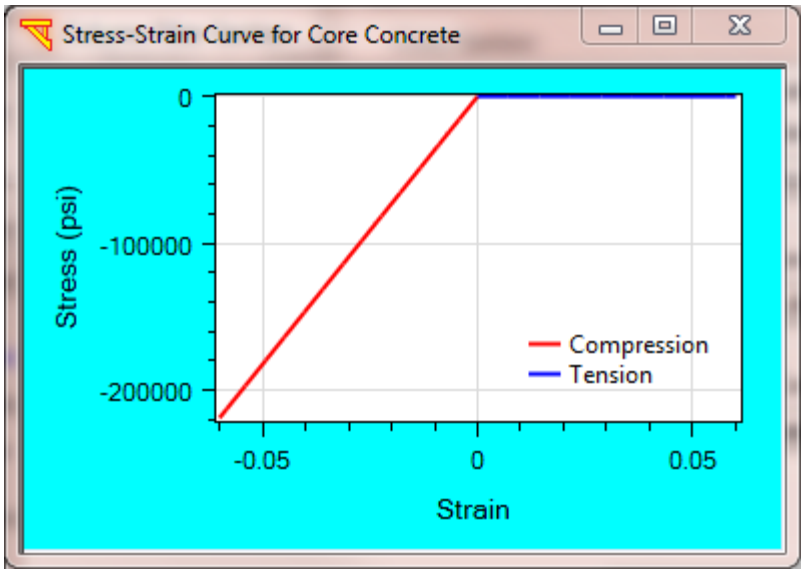


b)

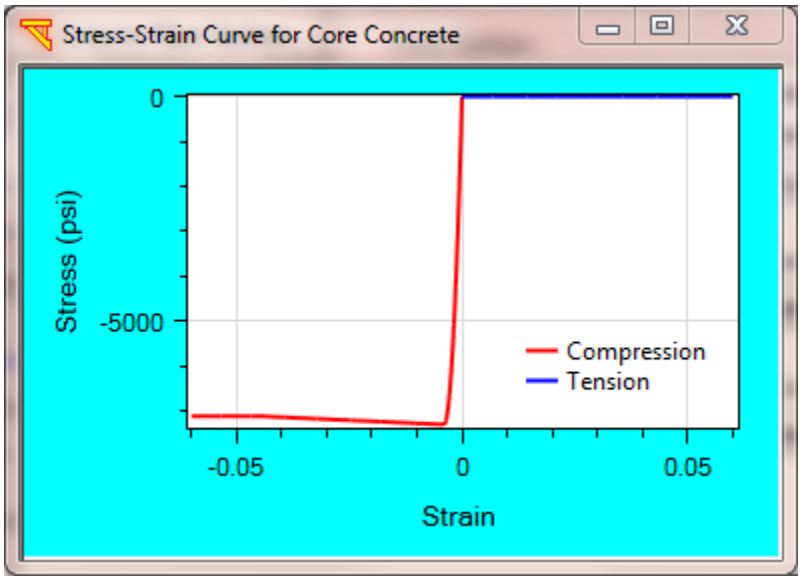


c)

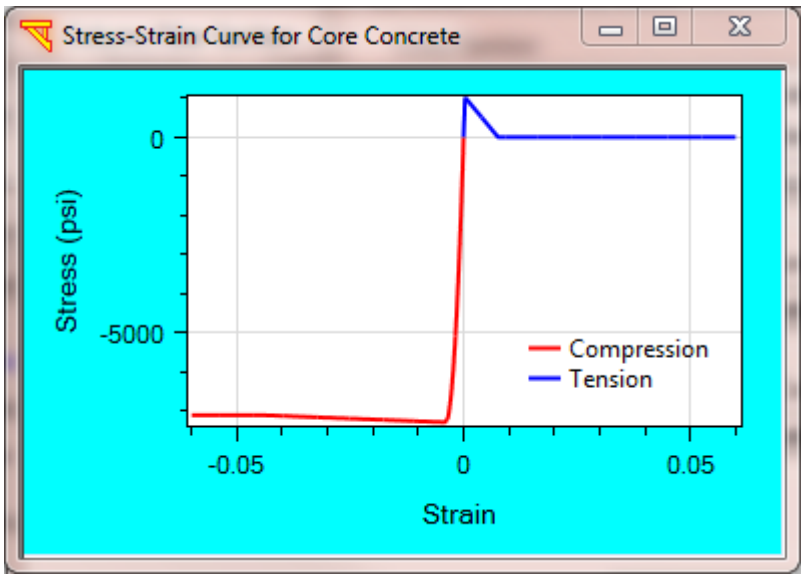
Fig. 31. Stress-strain curve for a steel material (default values employed): a) Steel01 (with a strain limit); b) Steel02 (with a strain limit); and c) ReinforcingSteel (with a strain limit)


Stress-Strain Curve for Core Concrete
0
-100000
-200000
Stress (psi)
-0.05
0
0.05
Strain
Compression
Tension

a)

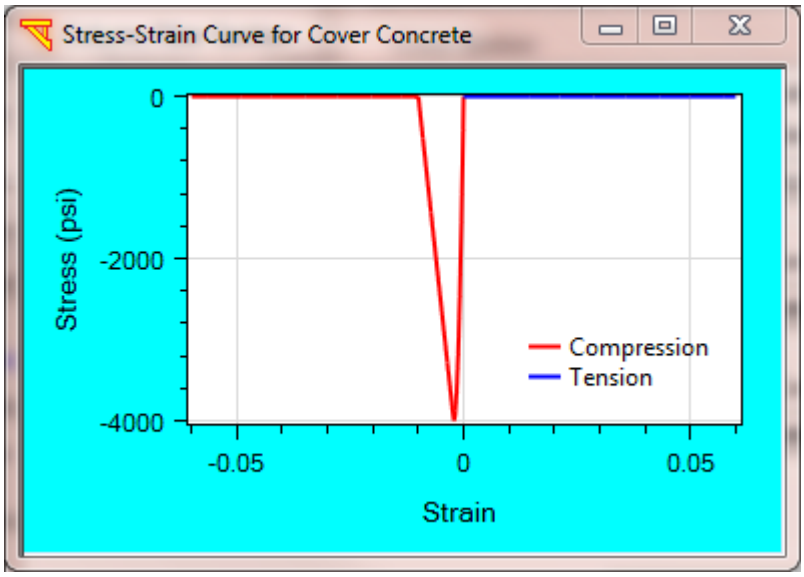

Stress-Strain Curve for Core Concrete
0
-5000
Stress (psi)
-0.05
0
0.05
Strain
Compression
Tension

b)

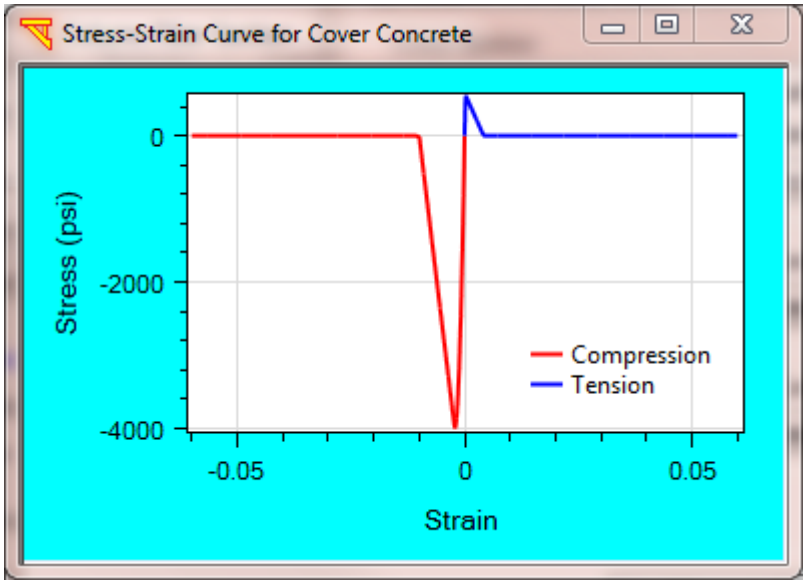


c)

Fig. 32. Stress-strain curve of the core concrete material (default values employed): a) Elastic-No Tension; b) Concrete01; and c) Concrete02



a)



b)

Fig. 33. Stress-strain curve of the cover concrete material (default values employed): a) Concrete01; and b) Concrete02

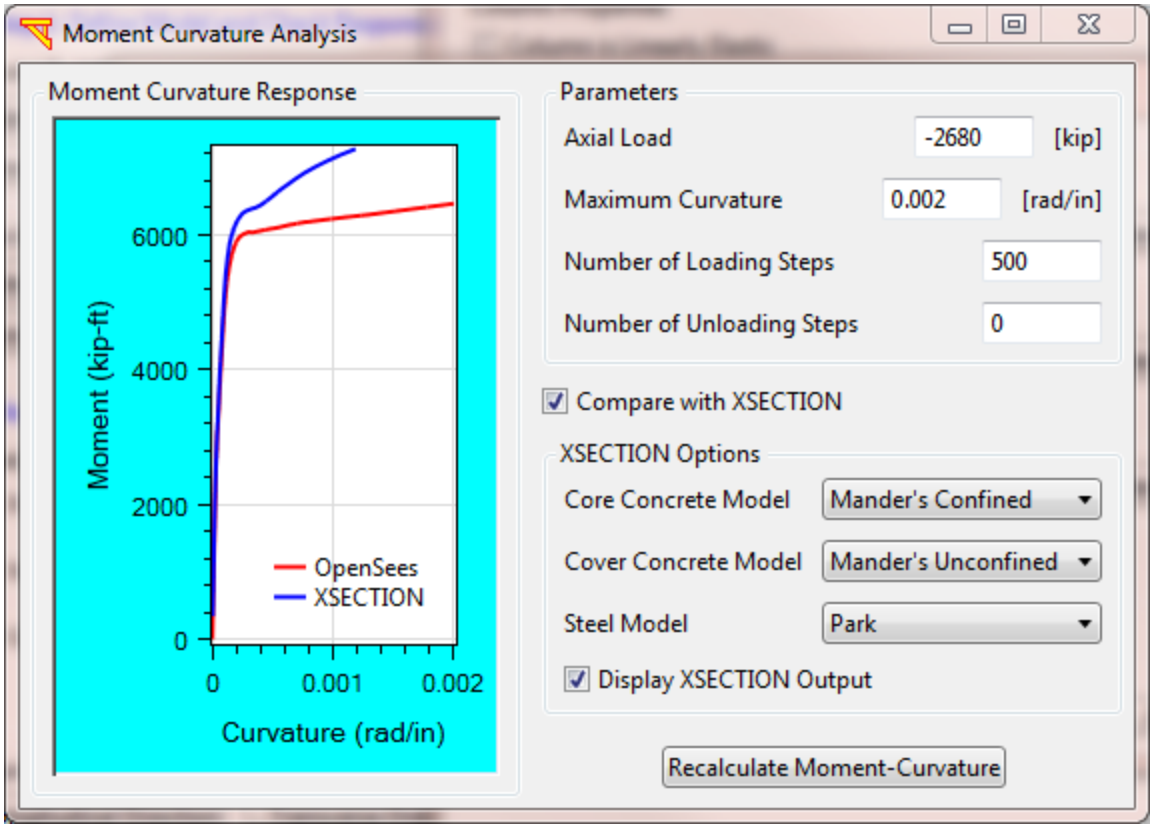


Fig. 34. Moment-curvature response for the column (with default steel and concrete parameters, and the deck weight 2680 kip applied at the column top

3.5 Foundation

3.5.1 Rigid Base

There are three types of foundations available (Fig. 35): **Rigid Base**, **Soil Springs** and **Foundation Matrix**. If Rigid Base is chosen, all column bases will be fixed (in 3 translational and 3 rotational directions). In that case, the "fixity" nodes of the abutment models are also fixed.

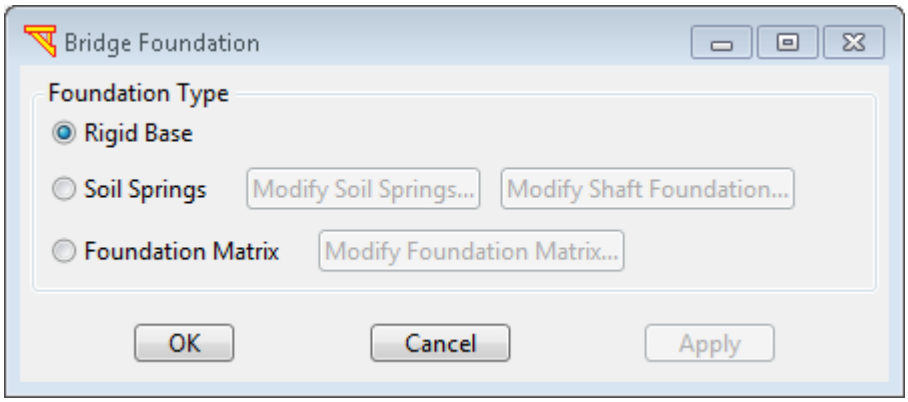


Fig. 35. Foundation types available in **MSBridge**

3.5.2 Soil Springs

To define soil springs, choose **Soil Springs** (Fig. 35) and then click **Modify Soil Springs** to define soil spring data or click **Modify Shaft Foundation** to define pile shaft data (Fig. 35). It is possible to include a shaft foundation at particular bents or abutments, simply check the box to turn off/on shaft foundation for each bent/abutment.

Parameters defining the pile foundation include (Fig. 36):

Pile Diameter: the diameter of the pile shaft (the cross section is assumed to be circular), which is 48 in by default.

Young's Modulus: Young's Modulus of the pile shaft. The foundation piles are assumed to remain linear.

Pile Group Layout (see Fig. 36). This option allows defining the numbers of pile as well as the spacing (in the bridge longitudinal and transverse directions). For now, this option is only available for both abutments. For a bent, one single pile is assumed.

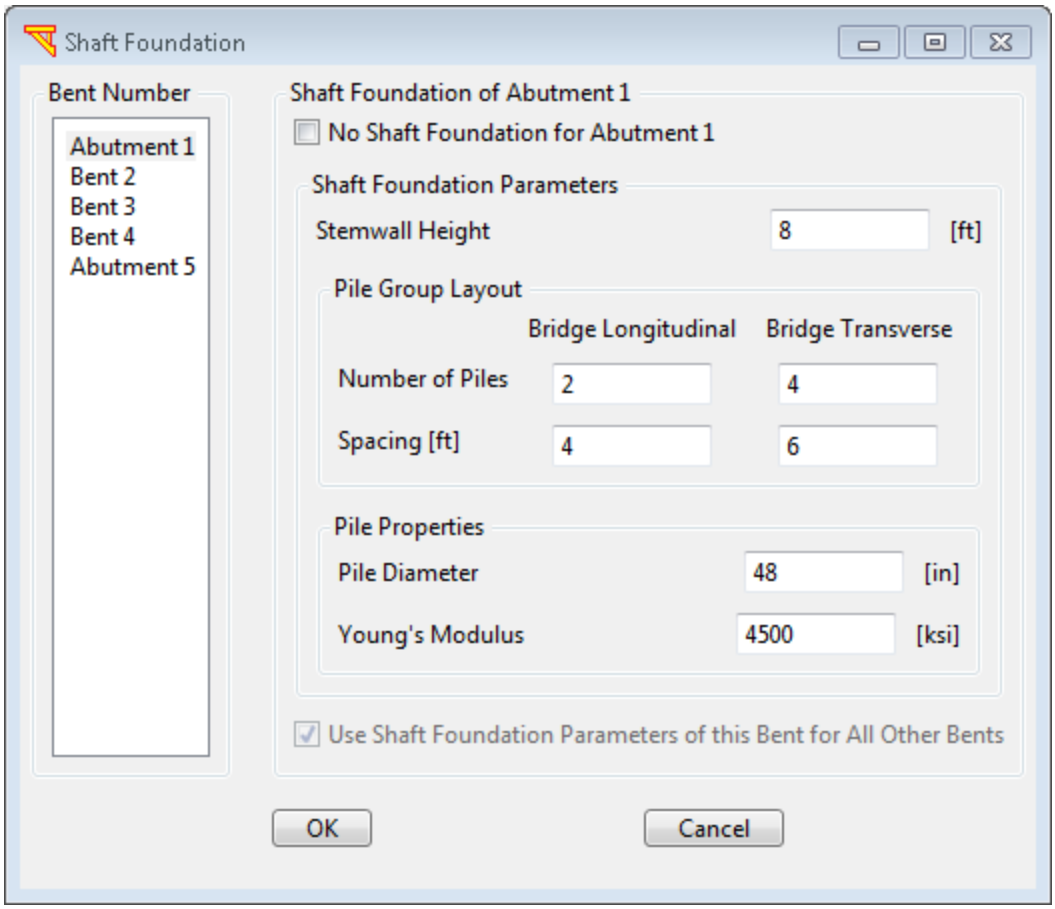


Fig. 36. Shaft foundation for abutments and bents

When implementing the soil springs for an abutment section consider Fig. 37 as a representation of the model. The “abutment nodes” are the same nodes that will be referred to in the abutment model section. These nodes are described as the “fixities” in each of the SDC models figures.

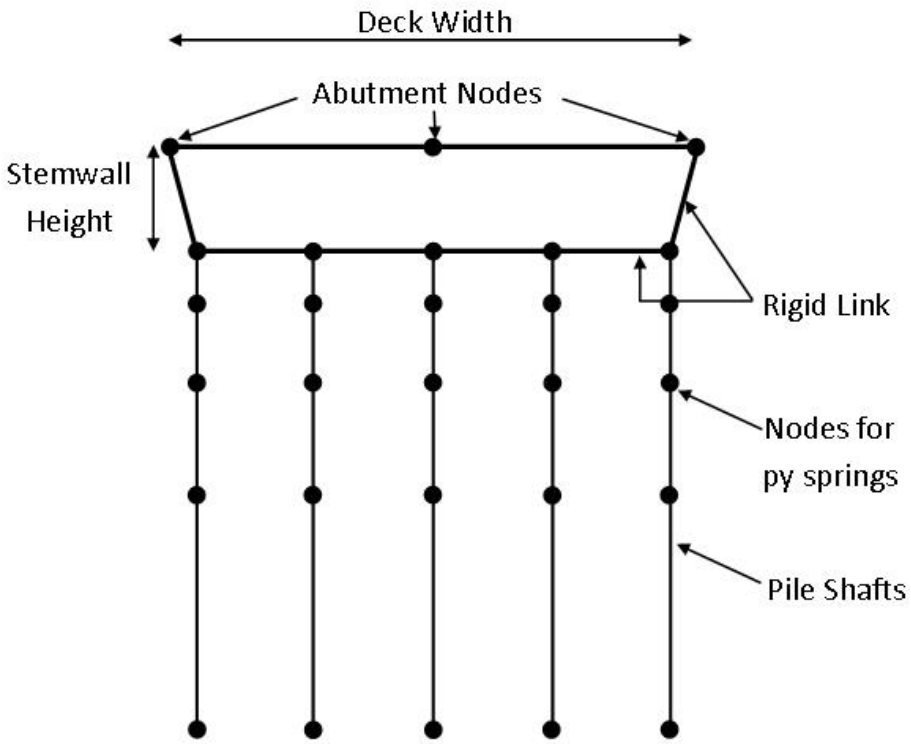


Fig. 37. Pile foundation model for abutments

Parameters defining soil springs are shown in Fig. 38. Two identical horizontal soil springs (one for the bridge longitudinal direction and the other one for the transverse direction) will be applied at each depth. Button **Insert Depth** inserts a depth after the current depth being highlighted. Button **Delete Depth** removes the current depth being highlighted (as well as the associated soil spring data).

To calculate the soil spring data based on p-y equations, click **Select from p-y Curves** (Fig. 38). For now, three types of soil p-y curves are available: **Soft Clay (Matlock), Stiff Clay with no Free Water (Reese), and Sand (Reese).** Fig. 39 show the calculated p-y curves for the above mentioned soil materials, respectively. The methods to calculate these p-y curves are based on the procedures described in the reference by Reese and van Impe (2001).

To use the soil spring data calculated based on p-y curves, click **OK** and then click **Yes**. The soil spring data chosen will replace the existing any soil spring data (Fig. 40). A sample bridge model with soil springs included is shown in Fig. 41.

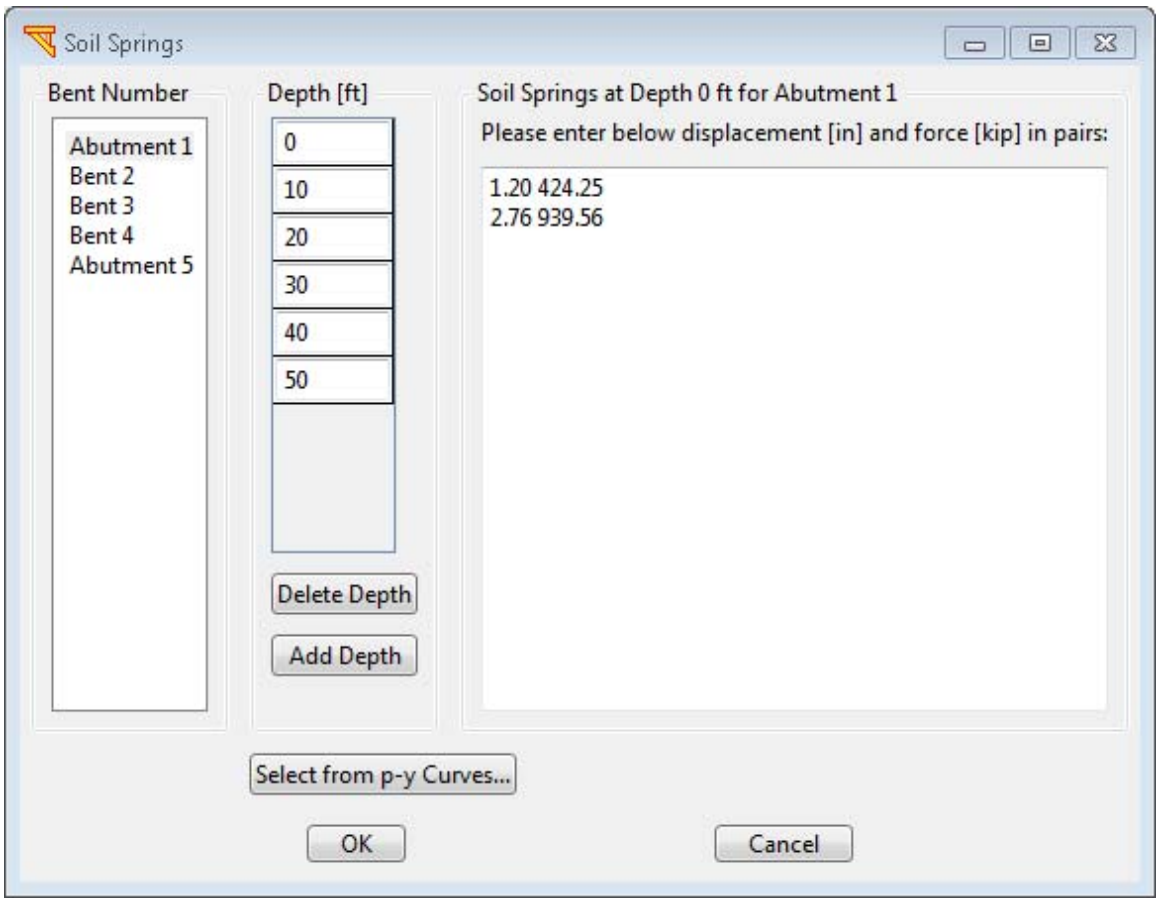
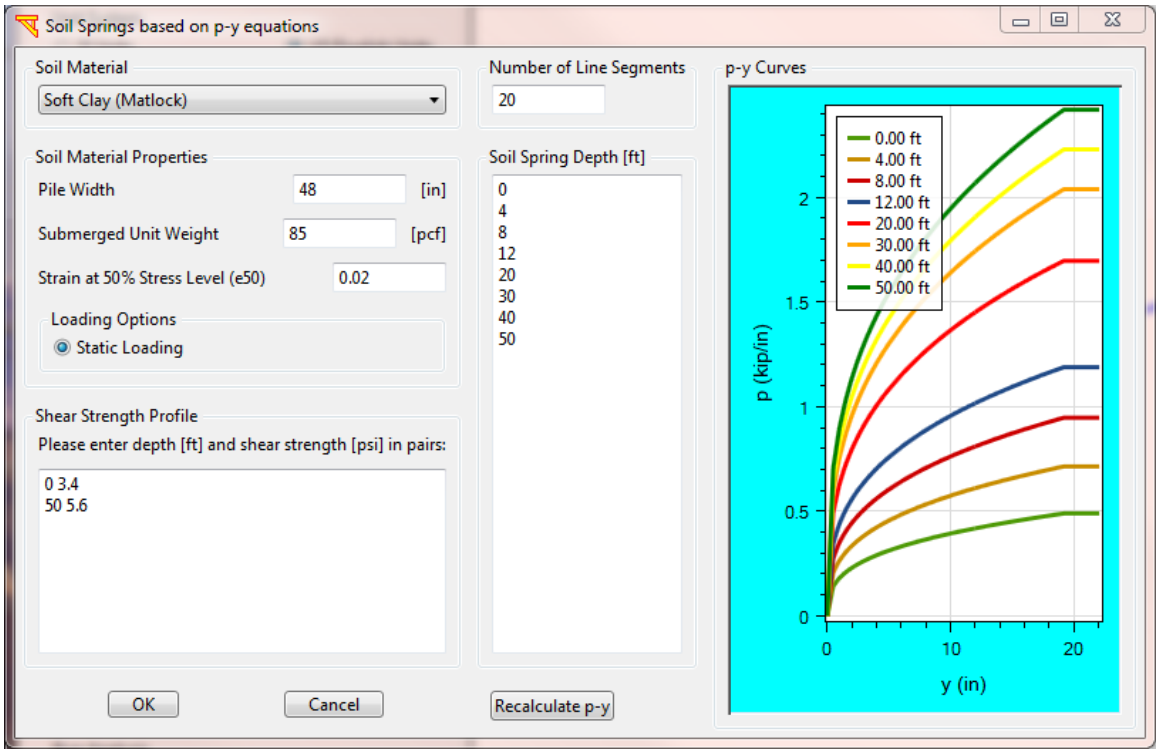
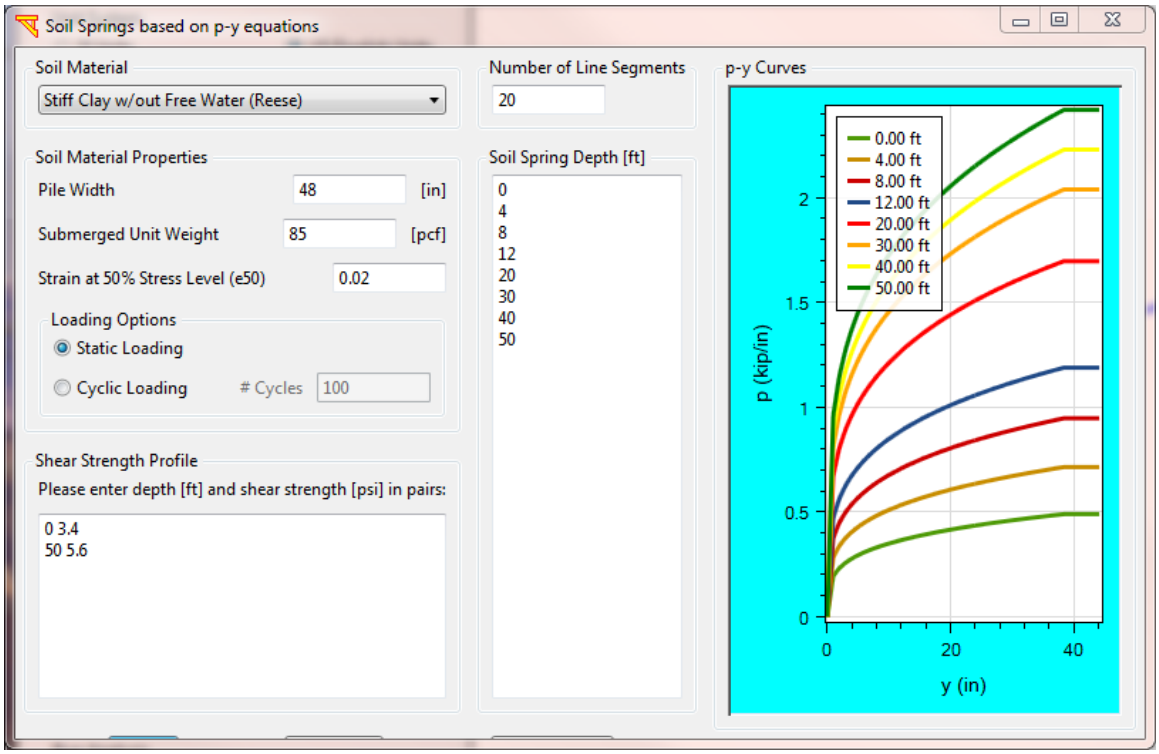


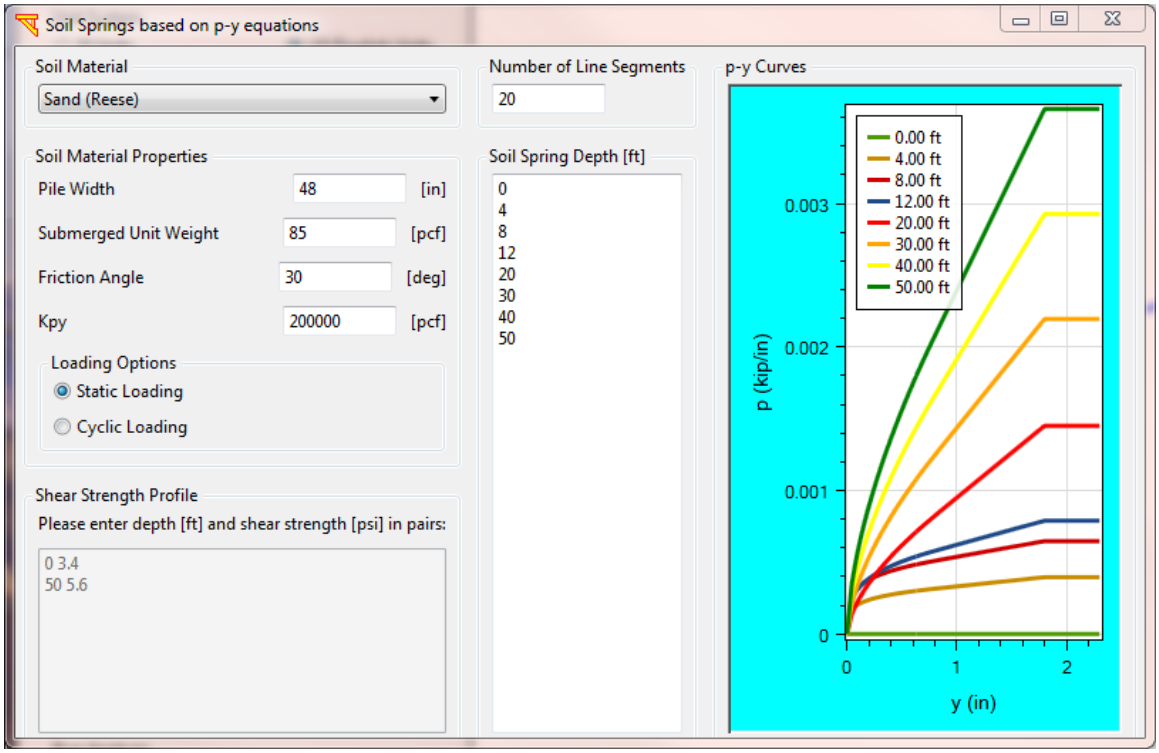
Fig. 38. Soil springs


Soil Springs based on p-y equations
Soil Material
Soft Clay (Matlock)
Number of Line Segments
20
p-y Curves
Soil Material Properties
Pile Width
48
[in]
Submerged Unit Weight
85
[pcf]
Strain at 50% Stress Level (e50)
0.02
Loading Options
Static Loading
Soil Spring Depth [ft]
0
4
8
12
20
30
40
50
Shear Strength Profile
Please enter depth [ft] and shear strength [psi] in pairs:
0 3.4
50 5.6
OK
Cancel
Recalculate p-y
0.00 ft
4.00 ft
8.00 ft
12.00 ft
20.00 ft
30.00 ft
40.00 ft
50.00 ft
p (kip/in)
y (in)

a)


Soil Springs based on p-y equations
Soil Material
Stiff Clay w/out Free Water (Reese)
Number of Line Segments
20
p-y Curves
Soil Material Properties
Pile Width
48
[in]
Submerged Unit Weight
85
[pcf]
Strain at 50% Stress Level (e50)
0.02
Loading Options
Static Loading
Cyclic Loading
Cycles
100
Soil Spring Depth [ft]
0
4
8
12
20
30
40
50
Shear Strength Profile
Please enter depth [ft] and shear strength [psi] in pairs:
0 3.4
50 5.6
0.00 ft
4.00 ft
8.00 ft
12.00 ft
20.00 ft
30.00 ft
40.00 ft
50.00 ft
p (kip/in)
y (in)

b)



c)

Fig. 39. Soil spring calculations based on p-y equations: a) Soft Clay (Matlock); b) Stiff Clay without Free Water (Reese); c) Sand (Reese)

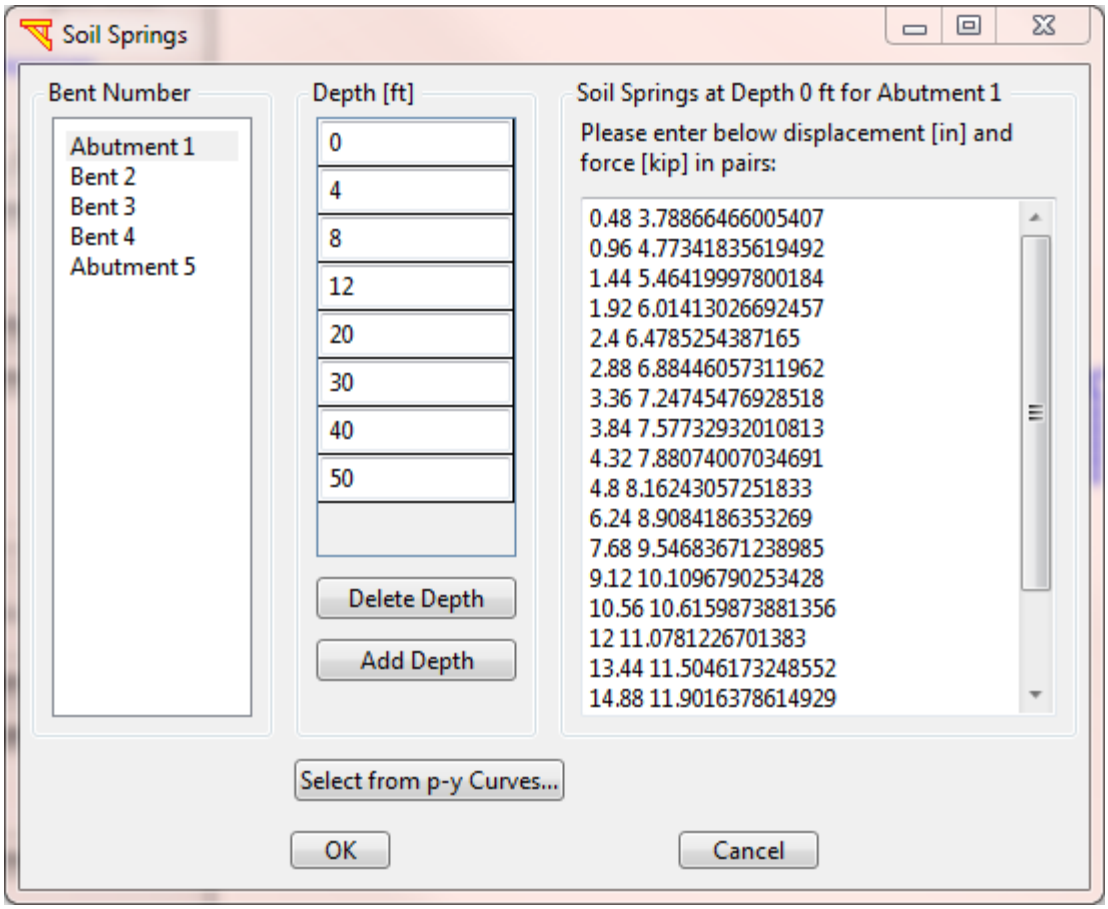


Fig. 40. Soil spring definition window after using the soil spring data calculated based on p-y equations

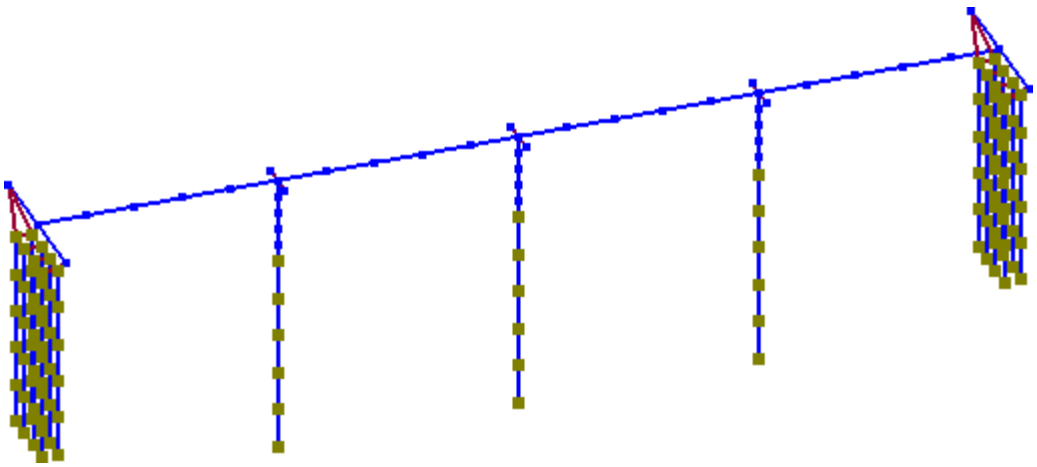


Fig. 41. Sample FE mesh of a bridge model with soil springs included

3.5.3 Foundation Matrix

The third foundation type available is **Foundation Matrix** (Fig. 35, Li and Conte 2013). In this method, the foundation (only for bent columns) is represented by the coupled foundation stiffness matrix (Lam and Martin 1986). Specifically, the stiffness of a single pile is represented by a 6 x 6 matrix representing stiffness associated with all six degrees of freedom at the pile head. The local coordination system employed for the foundation matrix is parallel to the global coordination system (Fig. 42).

To define foundation matrix, select **Foundation Matrix** and then click **Modify Foundation Matrix** (Fig. 35). Fig. 43 shows the window defining the foundation matrix for the column base of each bent. To apply the matrix defined for any bent for the remaining bents, select that bent in the **Bent No** box and then check **Use this Matrix for All Other Bents**.

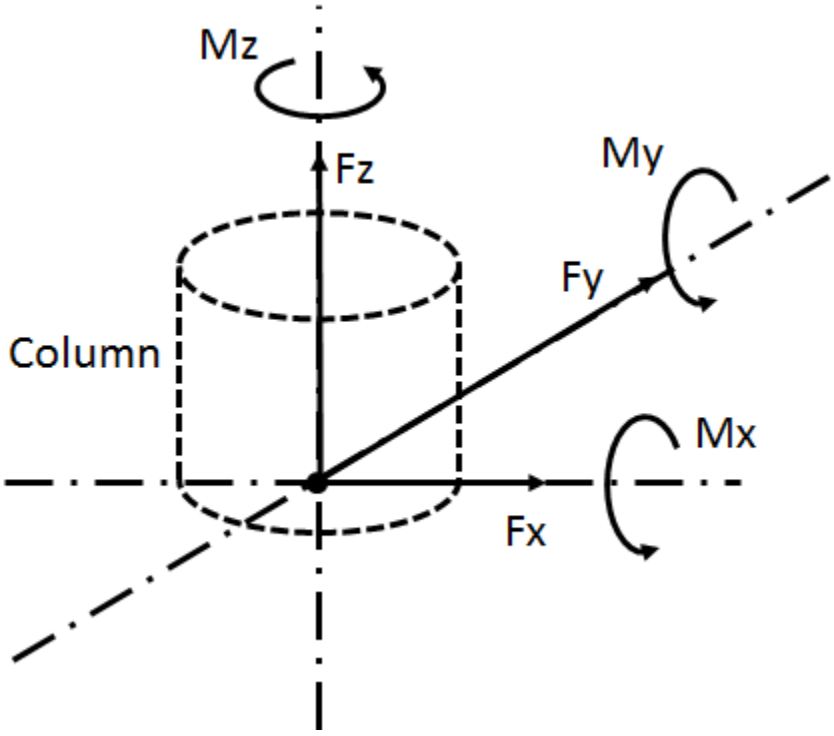


Fig. 42. Local coordination system for the foundation matrix

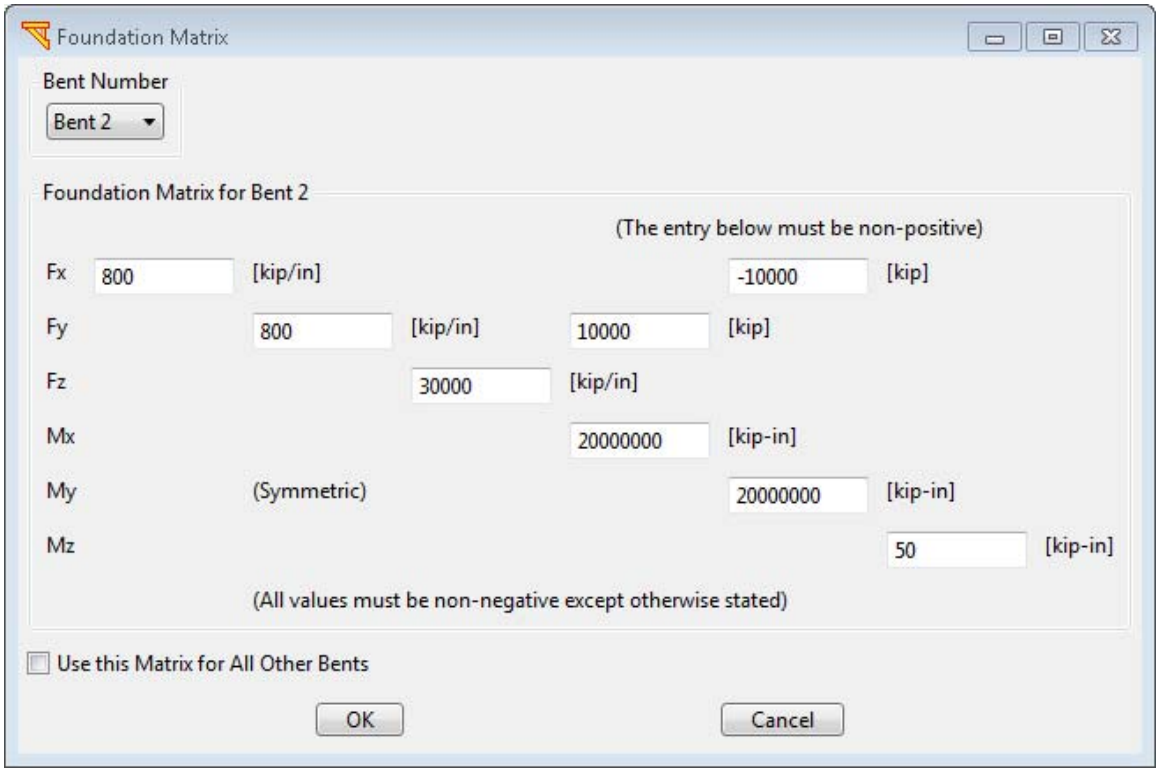


Fig. 43. Foundation matrix for each bent

3.6 Advanced Options

The advanced options in MSBridge include Deck Hinges, Isolation Bearings and Skew Angles. Click **Advanced** in Fig. 6 to include any of these options as shown in Fig. 44 into the bridge model.

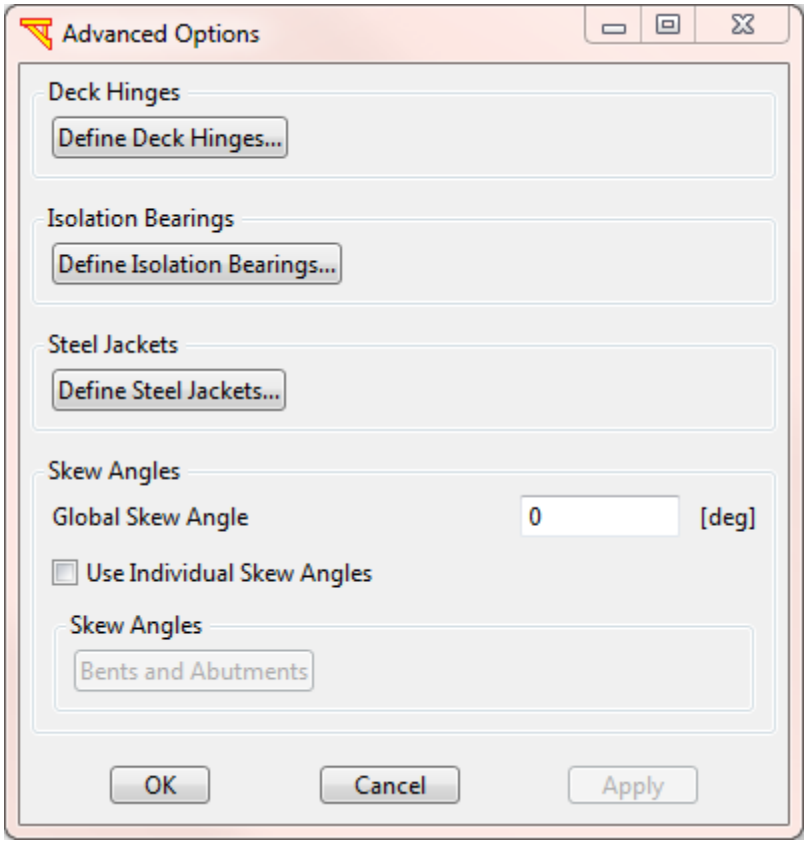


Fig. 44. Advanced options

3.6.1 Deck Hinges

To define deck hinges, click **Define Deck Hinges** in Fig. 44 and a window for defining deck hinge properties will appear (Fig. 45). A sample bridge model including 2 deck hinges is shown in
Fig. 46.

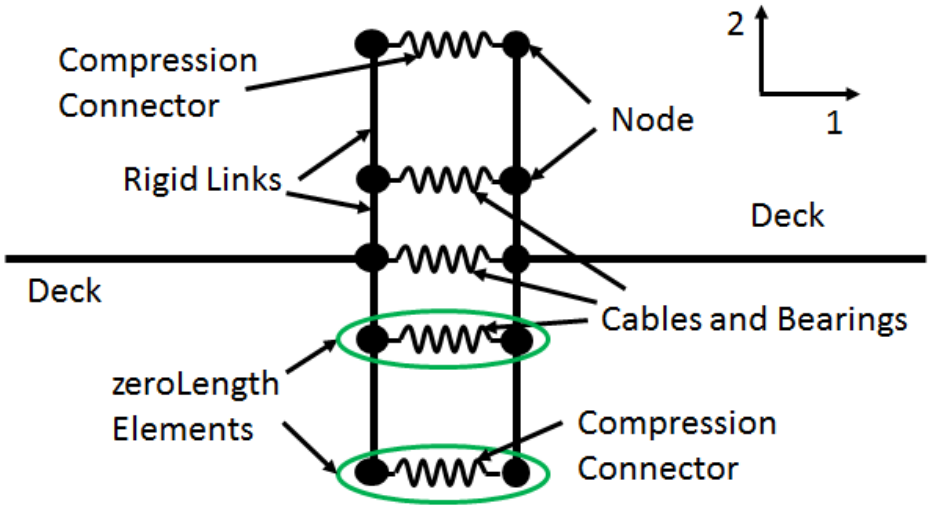


Fig. 47 shows the general scheme of a Deck Hinge, which consists of 2 compression connectors (located at both deck edges) and cables.

To activate/define a deck hinge, check the checkbox immediately prior to the **Hinge #** (e.g., Hinge 2).

> **Distance to Bent**: The distance to the nearest (left) bent. Foot and meter are used for English and SI units, respectively
> **Spacing**: The space between transverse left and right deck connectors. This space should usually approximately equal to the Deck Width.
> **Skew Angle**: The skew angle of the deck hinge. A zero skew angle means the deck hinge is perpendicular to the bridge deck direction
> **# of Cables**: The total number of cables of the deck hinge
> **Cable Spacing**: The spacing between cables. Symmetric layout of cables is assumed. Foot and meter are used for English and SI units, respectively

As shown in Fig. 47, zeroLength elements are used for cables and compression connectors. The bearing pads are included in the cables. For each zeroLength element, both nodes are interacted in the longitudinal direction (denoted as direction "1" in Fig. 47) but tied in the vertical direction "3" (not shown in Fig. 47) as well as the transverse direction (denoted as direction "2" in Fig. 47). The above conditions would force both sides of deck segments to move in the same plane. Note that the local coordinate system 1-2-3 may or may not coincide with the global coordinate system X-Y-Z (Fig. 1).

The default values of properties for the compression connectors, cables, bearing pads are also shown in Fig. 45.

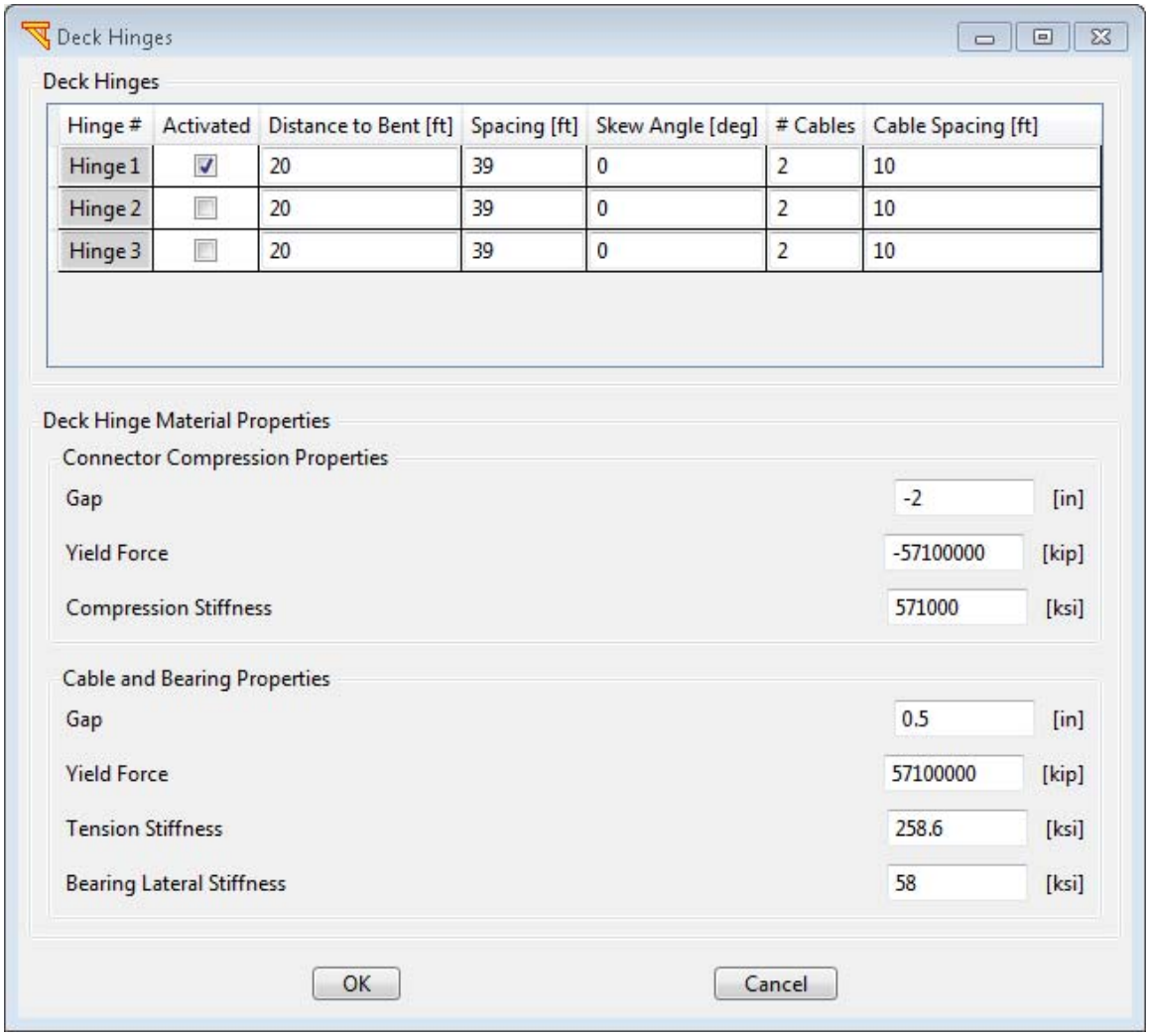

Deck Hinges
Deck Hinges
Hinge # | Activated | Distance to Bent [ft] | Spacing [ft] | Skew Angle [deg] | # Cables | Cable Spacing [ft]
Hinge 1 | ☑ | 20 | 39 | 0 | 2 | 10
Hinge 2 | ☐ | 20 | 39 | 0 | 2 | 10
Hinge 3 | ☐ | 20 | 39 | 0 | 2 | 10
Deck Hinge Material Properties
Connector Compression Properties
Gap -2 [in]
Yield Force -57100000 [kip]
Compression Stiffness 571000 [ksi]
Cable and Bearing Properties
Gap 0.5 [in]
Yield Force 57100000 [kip]
Tension Stiffness 258.6 [ksi]
Bearing Lateral Stiffness 58 [ksi]
OK
Cancel

Fig. 45. Definition of deck hinges

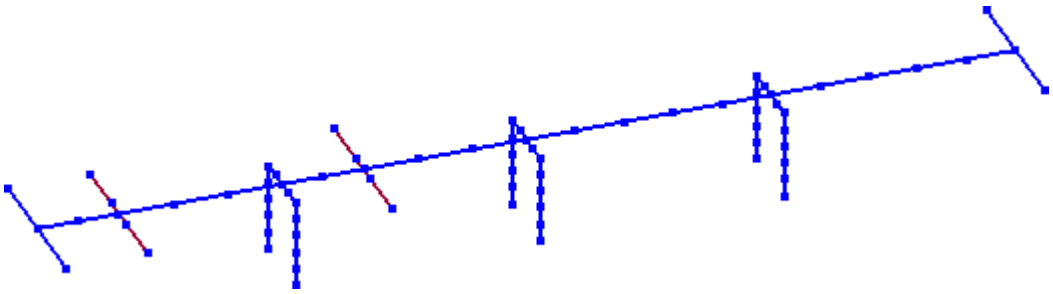


Fig. 46. FE mesh of a 4-span model with 2 deck hinges included

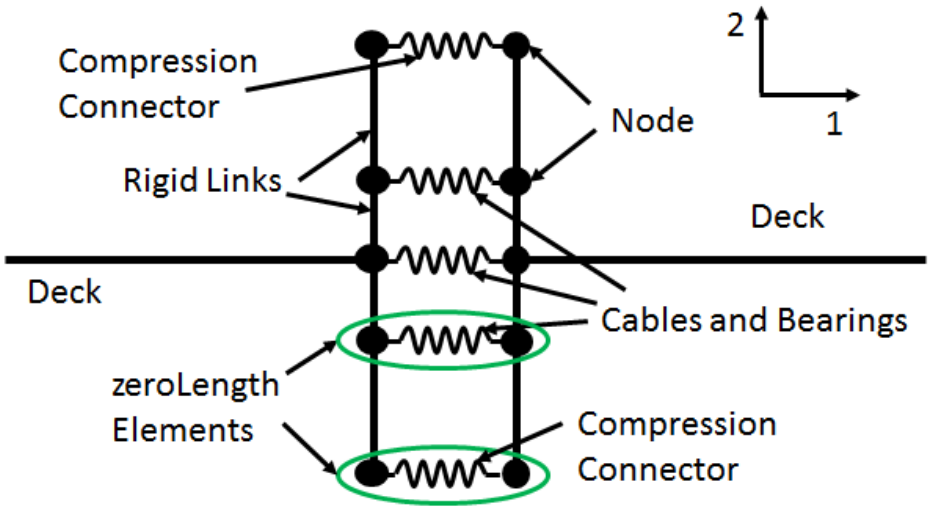


Fig. 47. OpenSees zeroLength elements for deck hinges (plan view)

3.6.2 Isolation Bearings

To define isolation bearings, click **Define Isolation Bearings** in Fig. 44 and a window for defining isolation bearing properties will appear (Fig. 48). A sample bridge model including 2 isolation bearings on each bent cap is displayed in Fig. 49.

To activate/define isolation bearings on a bent cap, check the checkbox immediately prior to the **Bent #** (e.g., Bent 2).

> **# Bearings**: The total number of isolation bearings implemented at the bent cap.
> **Spacing**: The spacing between isolation bearings. A symmetric layout of bearings is assumed.

The default values of material properties for the isolation bearings are also shown in Fig. 48. As shown in Fig. 50, zeroLength elements are used for the isolation bearings (Li and Conte, 2013). For each zeroLength element, the 2 nodes are interacted in both horizontal directions (denoted as directions "1"(not shown) and "2" in Fig. 50) but tied in the vertical direction "3" (Fig. 50). Note that the local coordinate system 1-2-3 may or may not coincide with the global coordinate system X-Y-Z (Fig. 1).

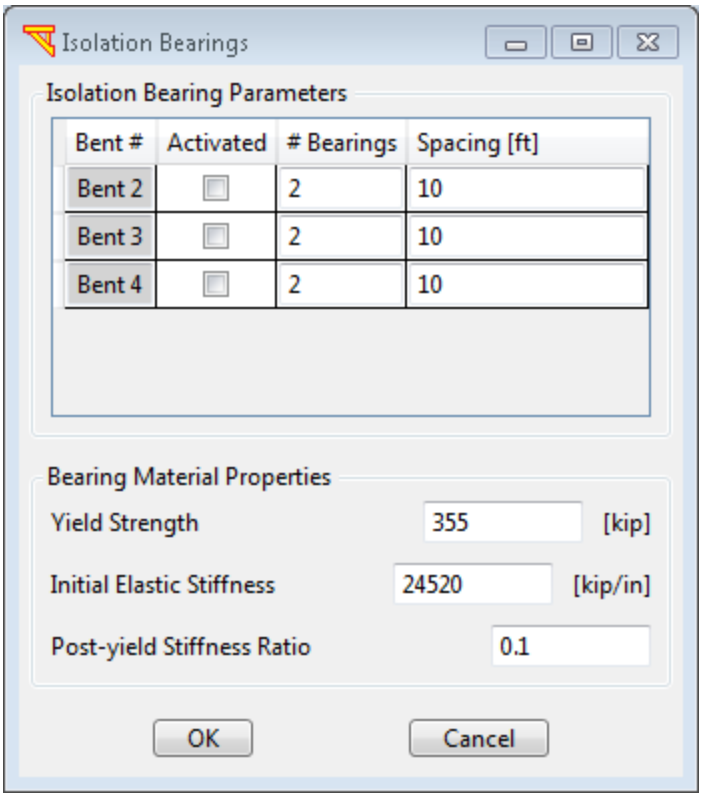


Fig. 48. Definition of isolation bearings

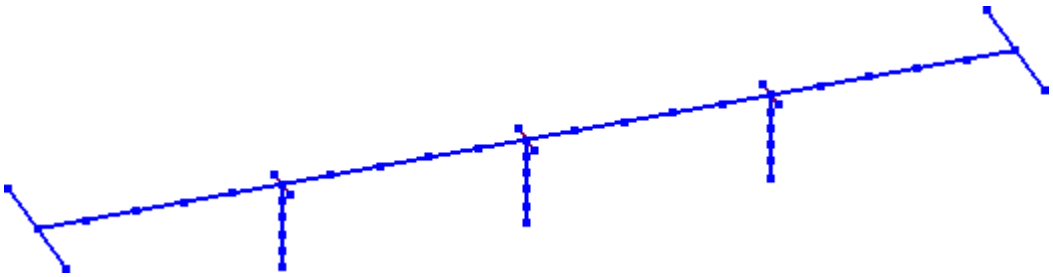


Fig. 49. FE mesh of a 4-span bridge model with 2 isolation bearings included on each bent cap

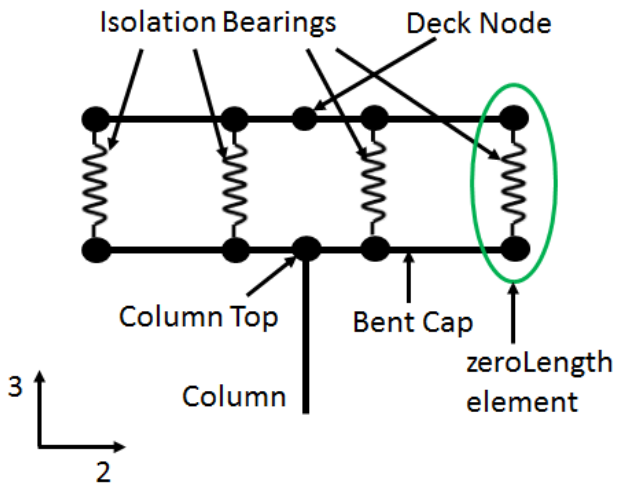


Fig. 50. OpenSees zeroLength elements for isolation bearings (side view of bent cap cut-plan)

3.6.3 Steel Jackets

To define steel jackets, click **Define Steel Jackets** in Fig. 44 and a window for defining steel jacket properties will appear (Fig. 51).

For now, the steel jacket option is only available to the circular column. To activate/define steel jacket for all columns for a bent, please nonzero values for the corresponding row (Fig. 51). In the case of partial length of steel jacket (Fig. 52), please specify enough number of elements for the column (since the equal size of elements is used for the columns within a bent, for now).

The steel properties used in the steel jacket implementation are the same as user defined properties for the steel reinforcement of the column shown in Fig. 27.

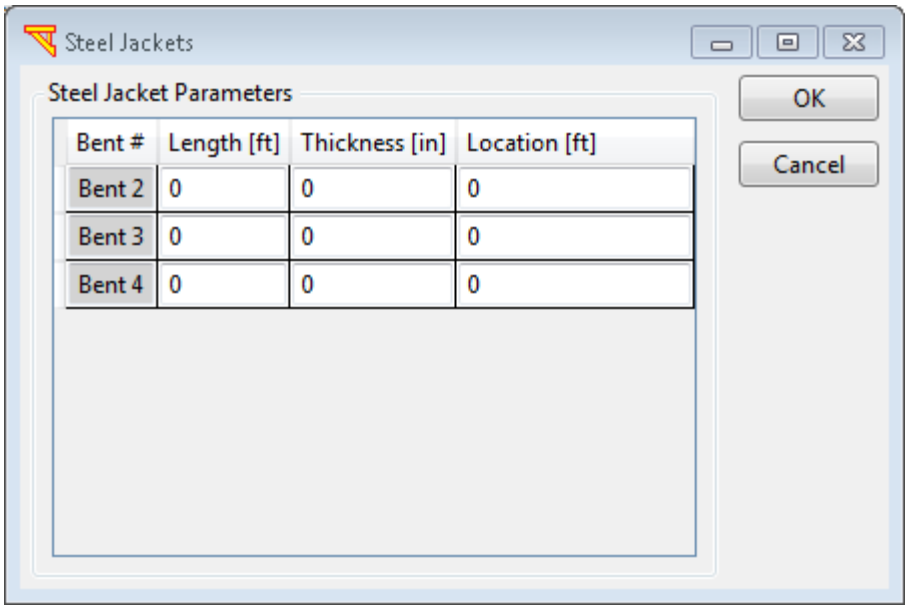


Fig. 51. Definition of steel jackets

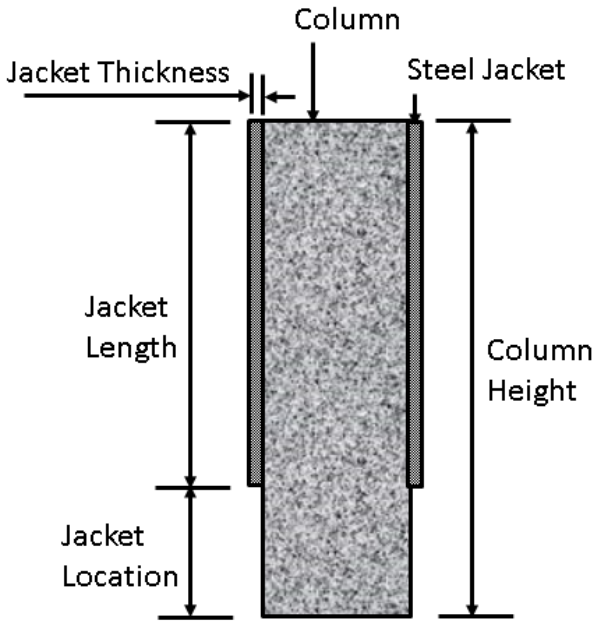
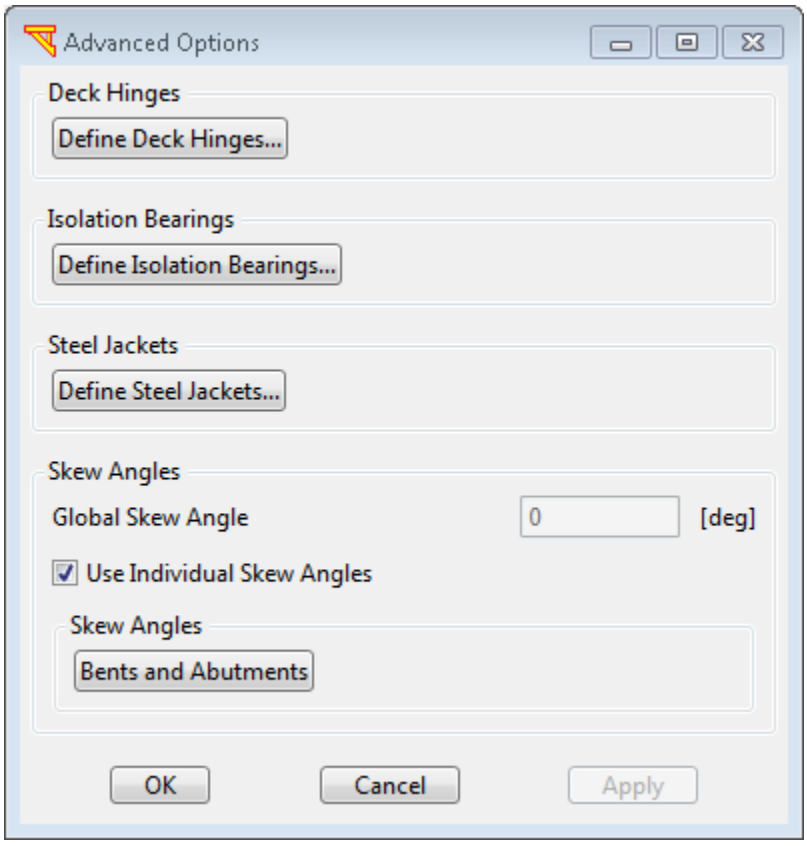


Fig. 52. Sketch of steel jacket

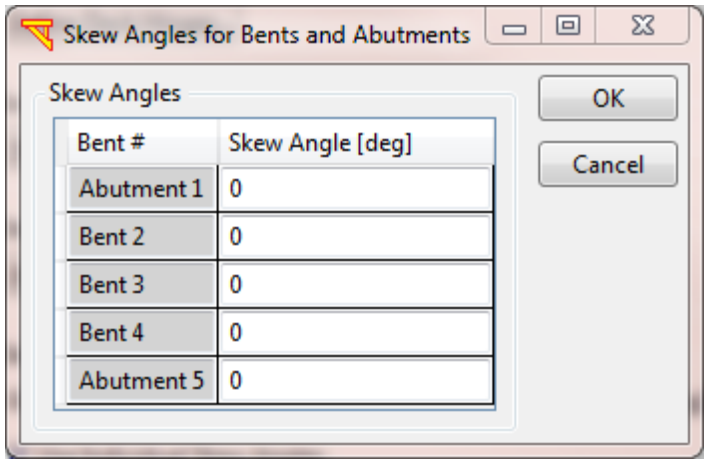
3.6.4 Skew Angles

The user can choose to use a single (global) skew angle or individual skew angles for abutments and bents. By default, a zero Global Skew Angle is assumed (Fig. 53). To define individual skew angles, check the checkbox **Use Individual Skew Angles**.

To define individual skew angles, click **Bents and Abutments** in Fig. 53a. A window for defining skew angle properties will appear (Fig. 53b).



a)



b)

Fig. 53. Definition of skew angles

3.7 Mesh Parameters

To change the number of beam-column elements for the bridge model, click **Mesh** in Fig. 6. Fig. 54 displays the **Mesh Parameters** window showing the default values. The number of beam-column elements for a deck segment (a span) must be least 2. And for the bent cap segment between columns, the number of elements must be even.

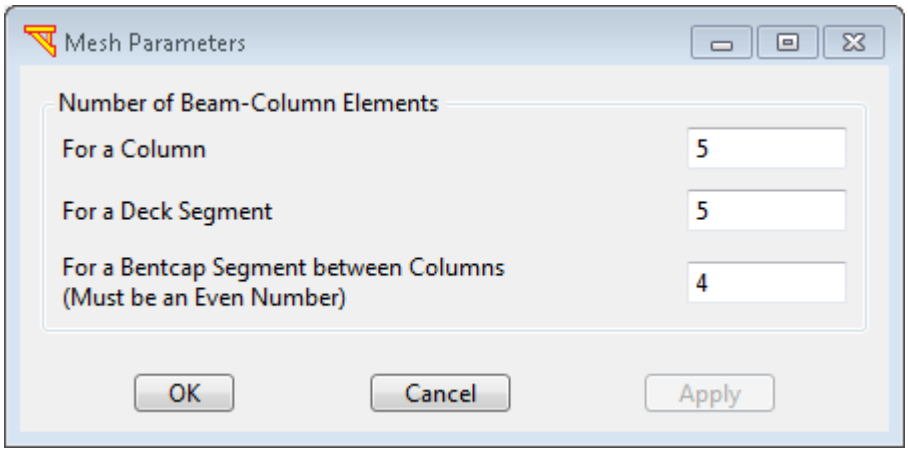


Fig. 54. Mesh parameters

4 Abutment Models

Abutment behavior, soil-structure interaction, and embankment flexibility have been found by post-earthquake reconnaissance reports to significantly influence the response of the entire bridge system under moderate to strong intensity ground motions. Specifically, for Ordinary Standard bridge structures in California with short spans and relatively high superstructure stiffness, the embankment mobilization and the inelastic behavior of the soil material under high shear deformation levels dominate the response of the bridge and the intermediate column bents (Kotsoglu and Pantazopoulou, 2006, and Shamsabadi et al. 2007, 2010). Seven abutment models have been implemented in **MSBridge**. The abutment models are defined as Elastic, Roller, SDC 2004, SDC 2010 Sand, SDC 2010 Clay, EPP-Gap and HFD abutment models.

To define an abutment model, click **Abutments** in Fig. 6. A window for defining an abutment model is shown in Fig. 55.

4.1 Elastic Abutment Model

The Elastic Abutment Model consists of a series of 6 elastic springs (3 translational and 3 rotational) at each node at the end of the bridge (Fig. 56). To choose the Elastic Abutment Model, select **Elastic** for the Model Type in Fig. 55 (Fig. 57). The main window to define the Elastic Abutment Model is shown in Fig. 58. By default, no additional rotational springs are specified, but can be added by the user.

As shown in Fig. 56 and Fig. 58, **MSBridge** allows the user to define multiple distributed springs (equal spacing within deck width). The values specified in Fig. 58 are the overall stiffness for each direction (translational or rotational). For the longitudinal direction (translational and rotational), each of the distributed (Elastic) springs carries its tributary amount.

e.g., Fig. 56 shows a case of 4 distributed springs. Each of the both end springs carries one-sixth of the load and each of the middle springs carries one-third (Fig. 56a). The vertical components (translational and rotational) are similar to the longitudinal ones. i.e., each of the distributed springs carries its tributary amount in the vertical direction. However, the transverse component is different: only the both end-springs carry the load. In other words, each of the end springs carries half of the load along the transverse direction (translational and rotational).

By default, the number of distributed springs is 2. In this case, these 2 springs are located at the both ends of the Rigid element (the length of which is equal to deck width) shown in Fig. 56. However, due to the coupling of the longitudinal, and vertical translational springs, the option of using a single node at each abutment is possible, this gives the user full control over the true rotational stiffness apart from the translational stiffness.

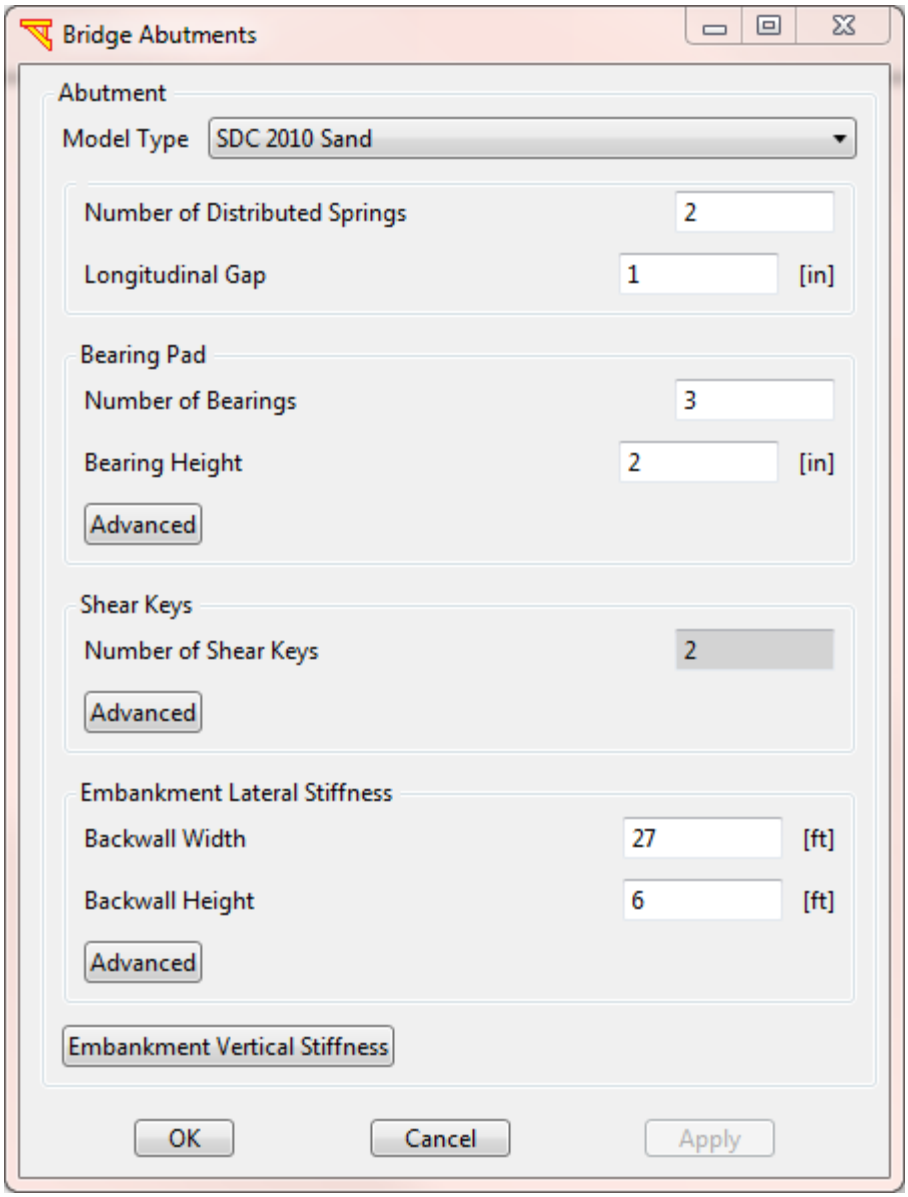
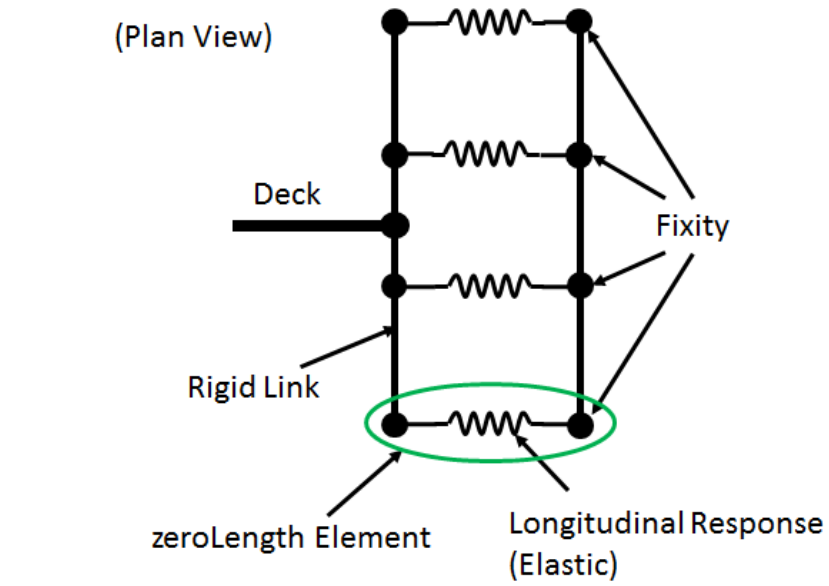
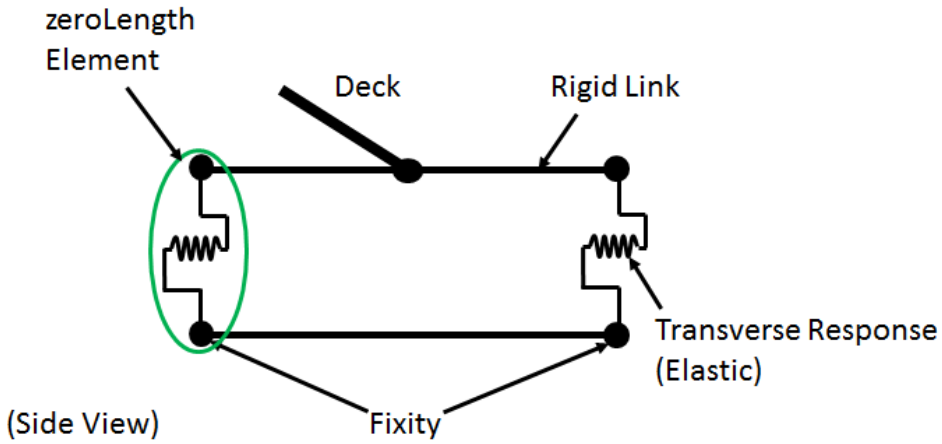


Fig. 55. Definition of an abutment model

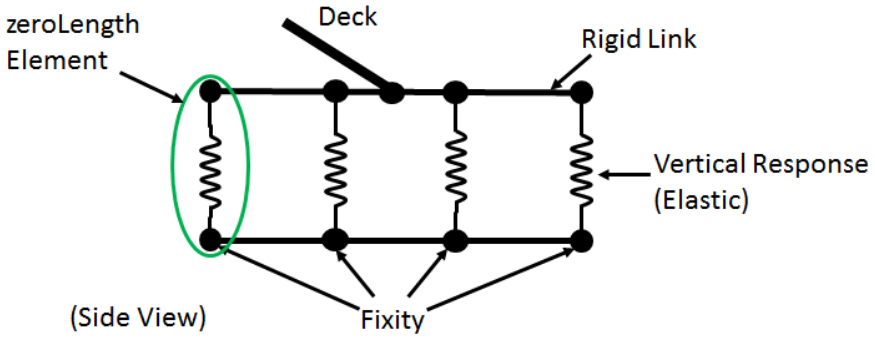
The abutment will be rotated counter-clockwise if the skew angle is positive (rotated clockwise if negative). Fig. 59 shows the direction of longitudinal springs in a curved bridge with a non-zero skew angle. Fig. 60 shows a bridge model with 5 distributed abutment springs and a non-zero skew angle.



a)



b)



c)

Fig. 56. General scheme of the Elastic Abutment Model: a) longitudinal component; b) transverse component; c) vertical component

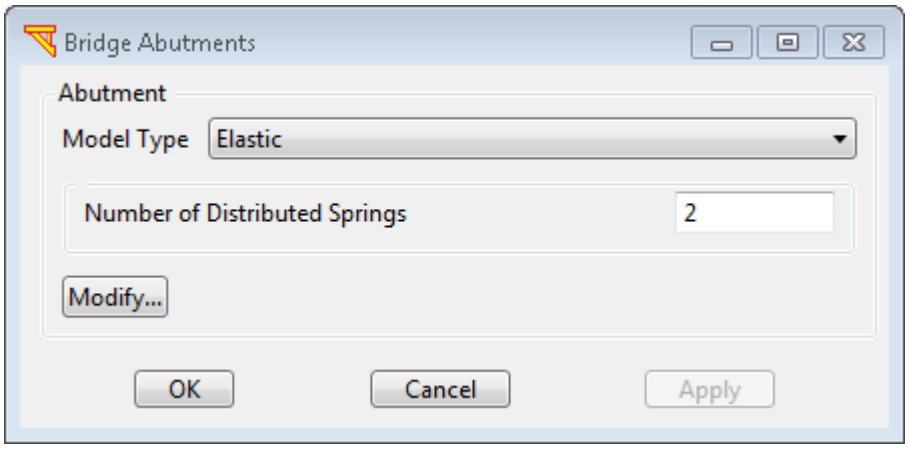


Fig. 57. Definition of the Elastic Abutment Model

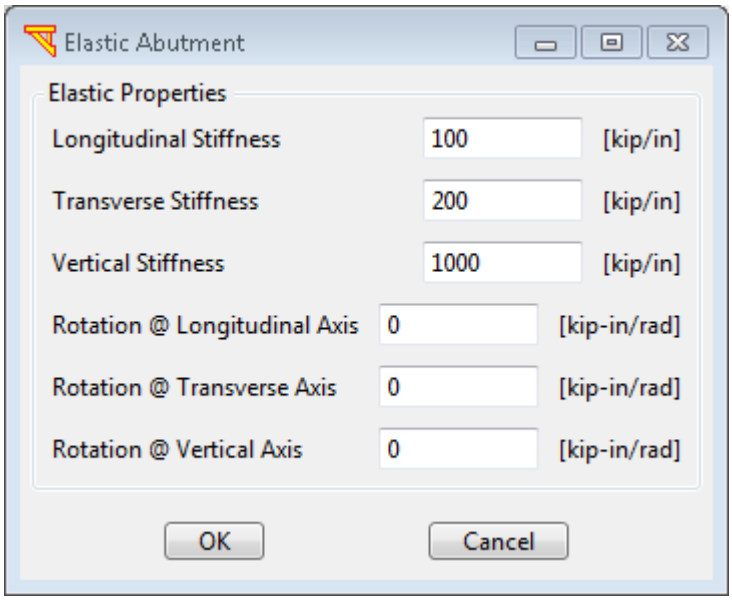
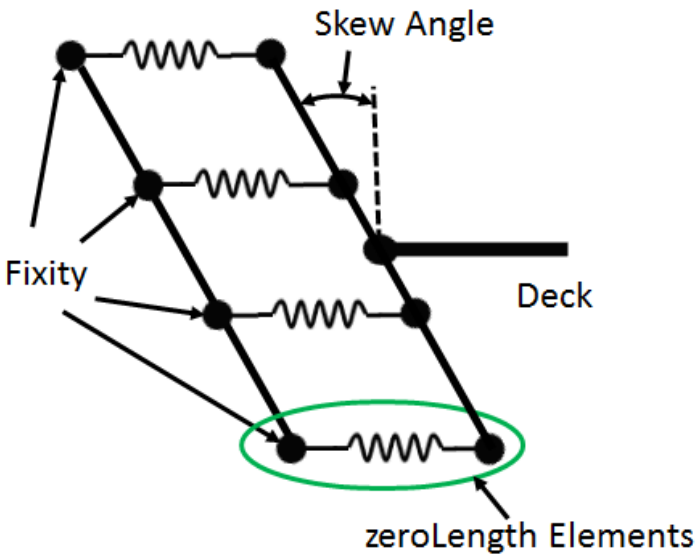
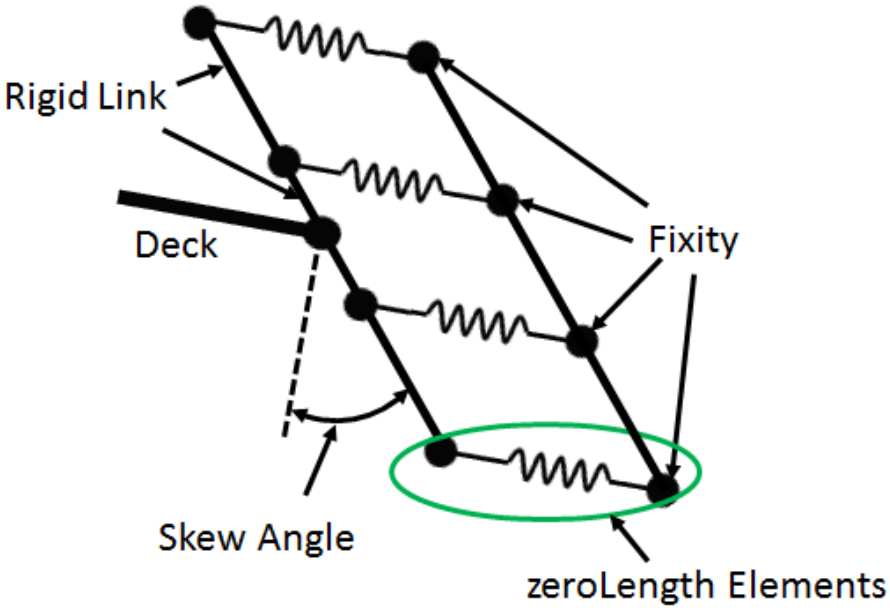


Fig. 58. Parameters of the Elastic Abutment Model

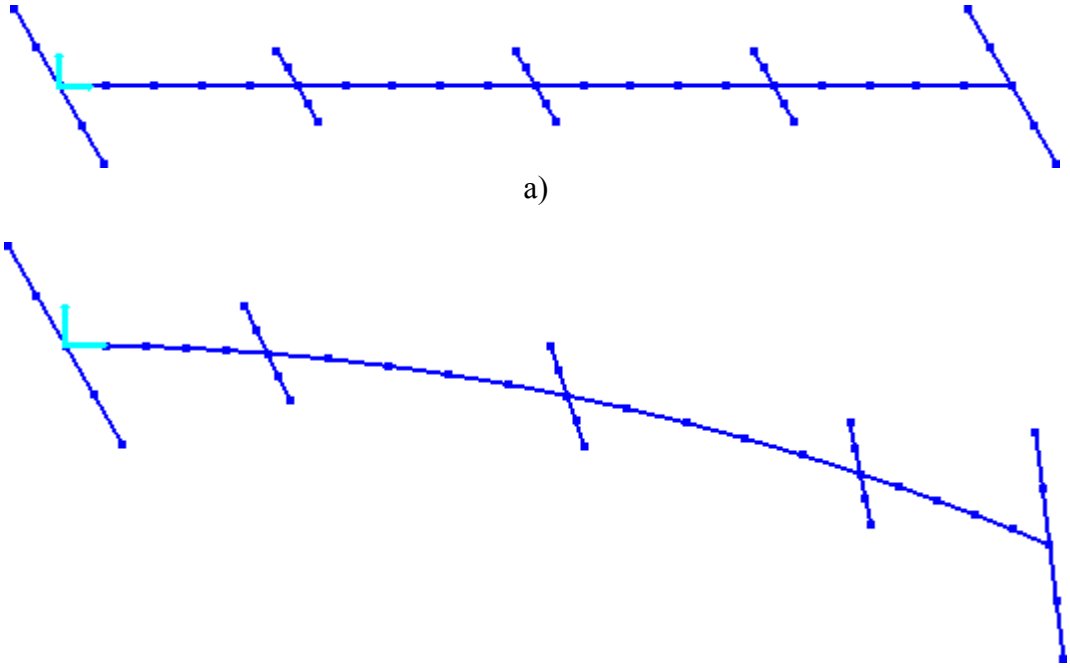


a)



b)

Fig. 59. Longitudinal components of the Elastic Abutment Model in a curved bridge: a) left abutment; b) right abutment



a)

b)

Fig. 60. Bridge model with multiple distributed springs and a positive skew angle: a) straight bridge; b) curved bridge

4.2 Roller Abutment Model

The Roller Abutment Model (Fig. 61) consists of rollers in the transverse and longitudinal directions, and a simple boundary condition module that applies single-point constraints against displacement in the vertical direction (i.e., bridge and abutment are rigidly connected in the vertical direction). These vertical restraints also provide a boundary that prevents rotation of the deck about its axis (torsion).

This model can be used to provide a lower-bound estimate of the longitudinal and transverse resistance of the bridge that may be displayed through a pushover analysis.

To choose the Roller Abutment Model, select **Roller** for the Model Type in Fig. 55 (and Fig. 62).

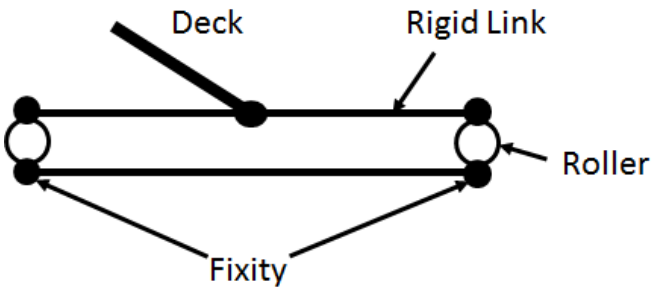


Fig. 61. General scheme of the Roller Abutment Model

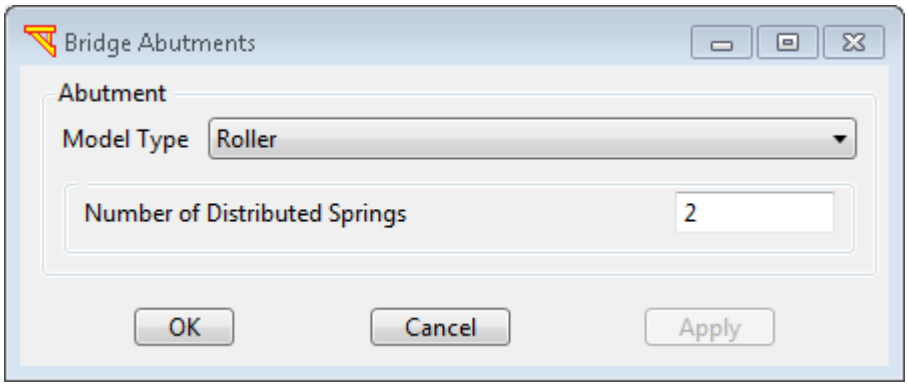


Fig. 62. Selection of the Roller Abutment Model

4.3 SDC 2004 Abutment Model

SDC 2004 Abutment Model was developed based on the Spring Abutment Model by Mackie and Stojadinovic (2006). This model includes sophisticated longitudinal, transverse, and vertical nonlinear abutment response. Detailed responses of the abutment model in the longitudinal, transverse, and vertical directions are described below.

4.3.1 Longitudinal Response

The longitudinal response is based on the system response of the elastomeric bearing pads, gap, abutment back wall, abutment piles, and soil backfill material. Prior to impact or gap closure, the superstructure forces are transmitted through the elastomeric bearing pads to the stem wall, and subsequently to the piles and backfill, in a series system. After gap closure, the superstructure bears directly on the abutment back wall and mobilizes the full passive backfill pressure. The detailed scheme of the longitudinal response is shown in Fig. 63a. The typical response of a bearing pad is shown in Fig. 63b. And the typical overall behavior is illustrated in Fig. 63c. The yield displacement of the bearings is

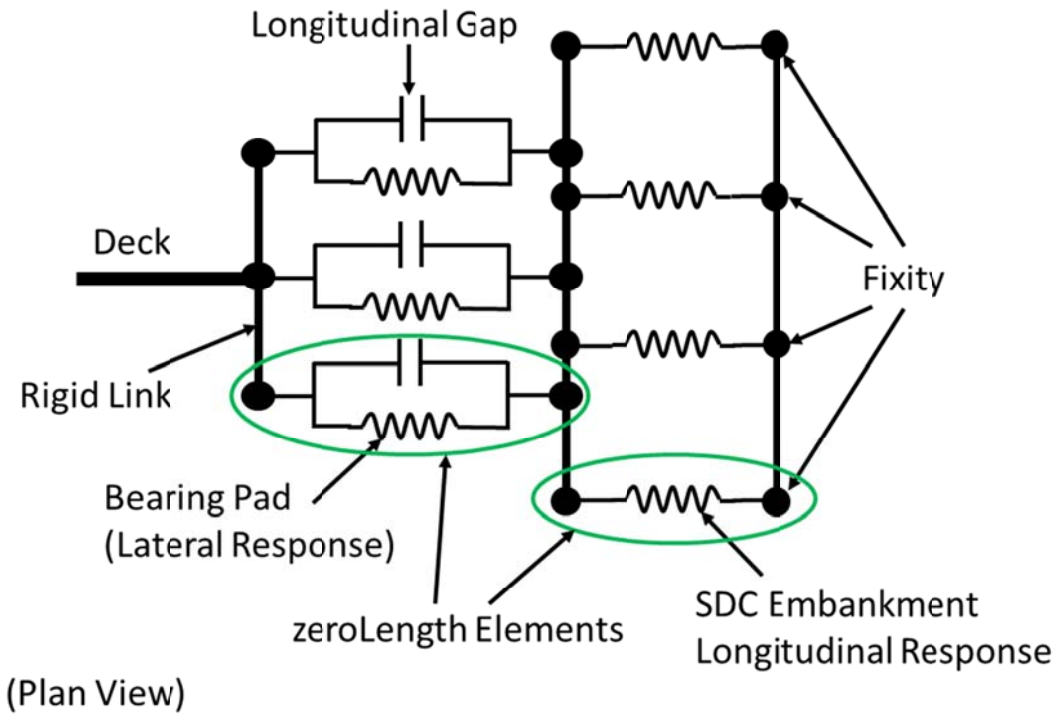
assumed to be at 150% of the shear strain. The longitudinal backfill, back wall, and pile system response are accounted for by a series of zero-length elements between rigid element 2 and the fixity (Fig. 63a). The abutment initial stiffness (Kabt) and ultimate passive pressure (Pabt) are obtained from equations 7.43 and 7.44 of SDC 2004. Fig. 64 shows the directions of zeroLength elements for a curved bridge with a skew angle.

Each bearing pad has a default height (h) of 0.0508 m (2 in) which can be modified by user and a side length (square) of 0.508 m (20 in). The properties of a bearing pad are listed in Table 4.

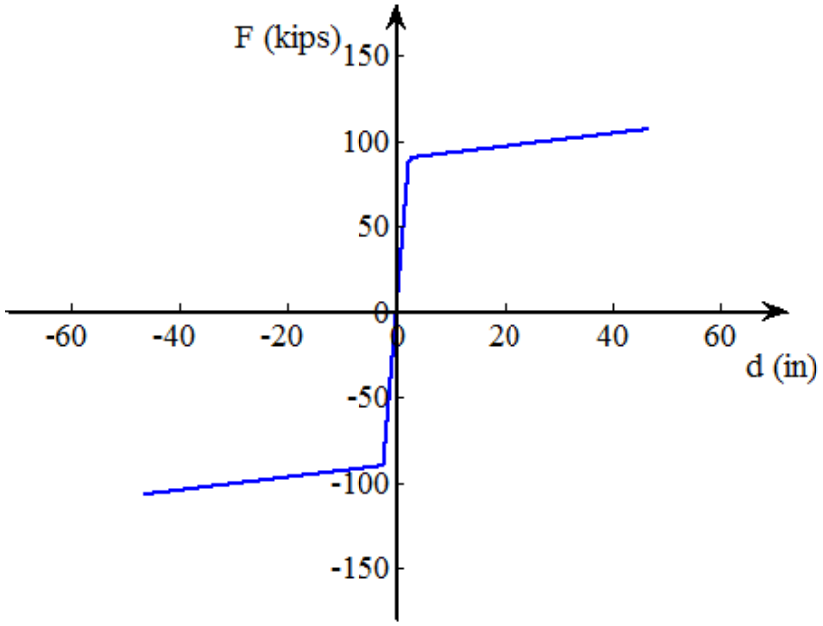
The abutment is assumed to have a nominal mass proportional to the superstructure dead load at the abutment, including a contribution from structural concrete as well as the participating soil mass. An average of the embankment lengths obtained from Zhang and Makris (2002) and Werner (1994) is included in the calculation of the participating mass due to the embankment of the abutment. The user can modify the lumped mass through the soil mass. For design purposes, this lumped mass can be ignored and set to be zero.

Table 4. Geometric and Material Properties of a Bearing Pad

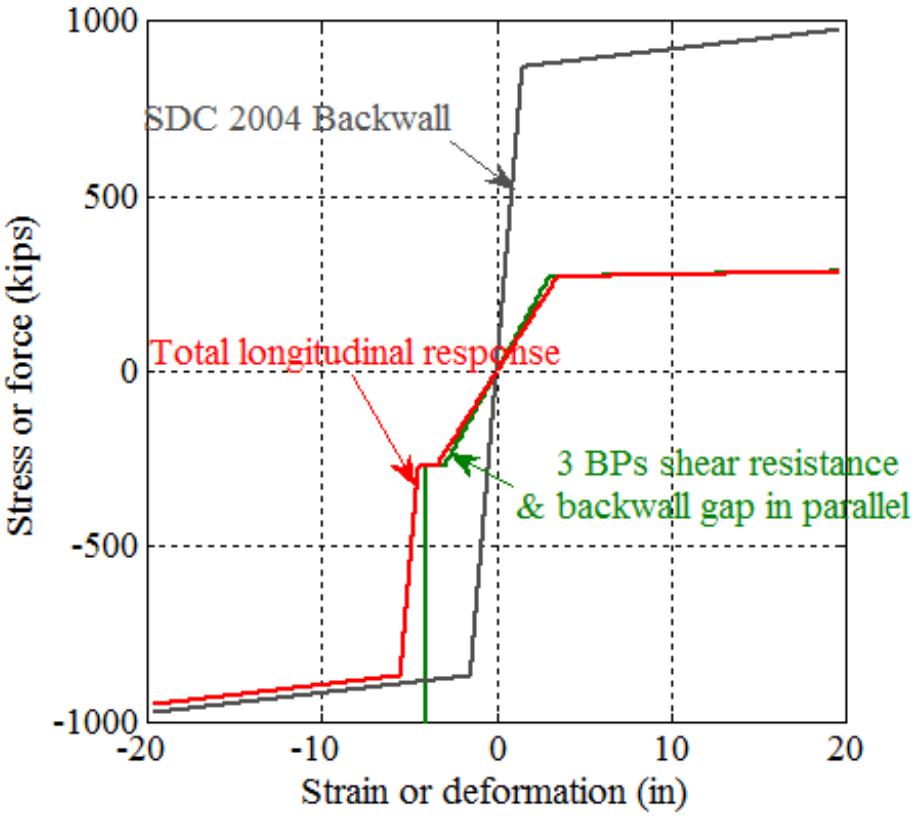
Shear Modulus G	1034.2 kPa (0.15 ksi)
Young's Modulus E	34473.8 kPa (5 ksi)
Yield Displacement	150% shear strain
Lateral Stiffness	$\frac{GA}{h}$ (where A is the cross-section area and h is the height)
Longitudinal gap	4 in
hardening ratio	1%
Vertical Stiffness	$\frac{EA}{h}$
Vertical Tearing Stress	15513.2 kPa (2.25 ksi)
Longitudinal gap	2 in


Longitudinal Gap
Deck
Fixity
Rigid Link
Bearing Pad
(Lateral Response)
zeroLength Elements
SDC Embankment
Longitudinal Response
(Plan View)

a)


F (kips)
150
100
50
0
-50
-100
-150
-60
-40
-20
0
20
40
60
d (in)

b)



c)

Fig. 63. Longitudinal response of the SDC 2004 Abutment Model: a) general scheme; b) longitudinal response of a bearing pad; c) total longitudinal response

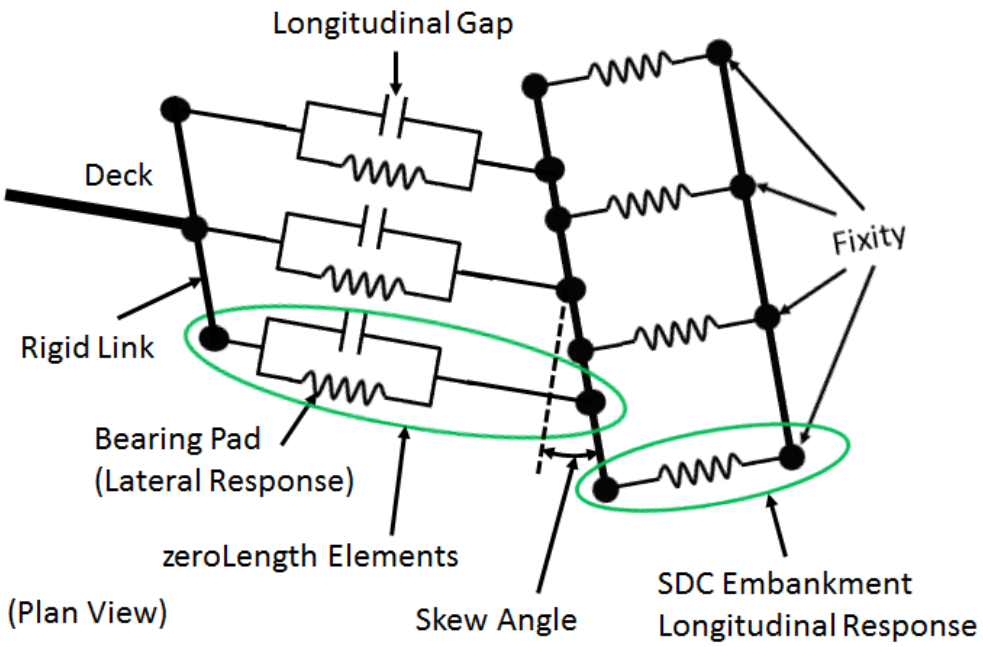


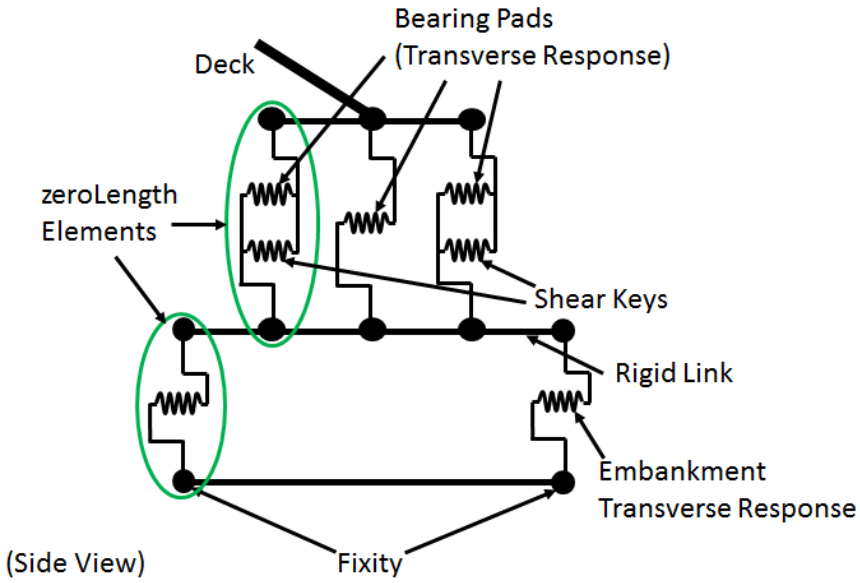
Fig. 64. Longitudinal response of the SDC 2004 Abutment Model for a curved bridge with a (positive) skew angle

4.3.2 Transverse Response

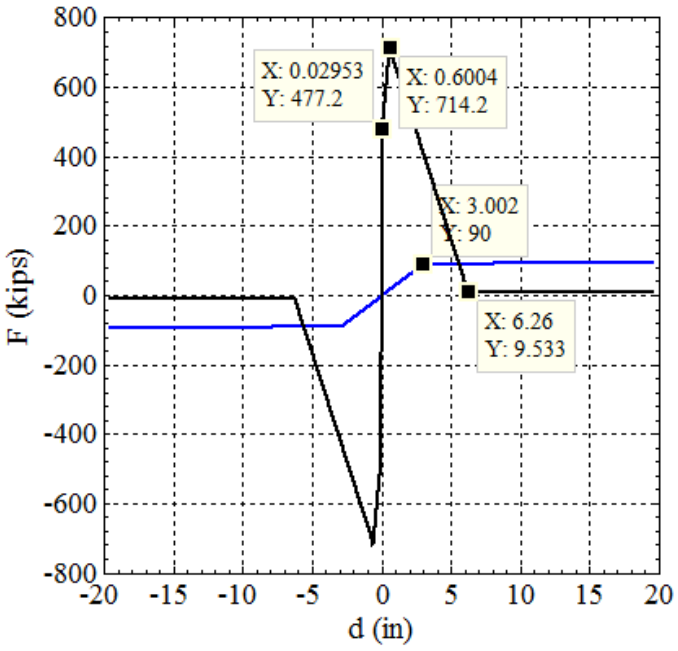
The transverse response is based on the system response of the elastomeric bearing pads, exterior concrete shear keys, abutment piles, wing walls, and backfill material. The bearing pad model discussed above is used with uncoupled behavior with respect to the longitudinal direction. The constitutive model of the exterior shear keys is derived from experimental tests (Megally et al., 2003). Properties (yield and ultimate stresses) of shear keys depend on ultimate capacity of the bridge which is defined as 30 percent of dead load at abutment.

The detailed scheme of the transverse response is shown in Fig. 65a. The typical response of a bearing pad and a shear key is shown in Fig. 65b. And the typical overall behavior of the transverse response is illustrated in Fig. 65c. The superstructure forces are transmitted through the parallel system of bearing pads and shear keys (T_1) to the embankment (T_2) in series. The ultimate shear key strength is assumed to be 30% of the superstructure dead load, according to equation 7.47 of SDC 2004. A hysteretic material with trilinear response backbone curve is used with two hardening and one softening stiffness values. The initial stiffness is a series-system stiffness of the shear and flexural response of a concrete cantilever with shear key dimensions (16849 ksi). The hardening and softening branches are assumed to have magnitudes of 2.5% of the initial stiffness. The transverse stiffness and strength of the backfill, wing wall and pile system is calculated using a modification of the SDC procedure for the longitudinal direction.

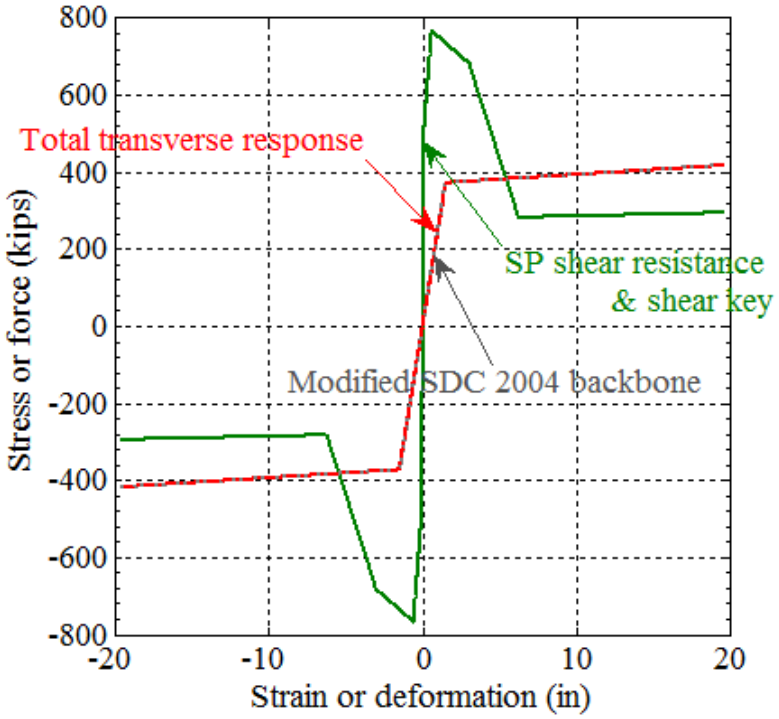
Wingwall effectiveness (CL) and participation coefficients (CW) of 2/3 and 4/3 are used, according to Maroney and Chai (1994). The abutment stiffness (Kabt) and back wall strength (Pbw) obtained for the longitudinal direction from Section 7.8 of SDC 2004 are modified using the above coefficients. The wing wall length can be assumed 1/2–1/3 of the back wall length. The bearing pads and shear keys are assumed to act in parallel. Combined bearing pad- shear key system acts in series with the transverse abutment stiffness and strength.



a)



b)



c)

Fig. 65. Transverse response of the SDC 2004 Abutment Model: a) general scheme; b) response of a bearing pad and shear keys (curve with a higher peak value is the shear key response); c) total transverse response

4.3.3 Vertical Response

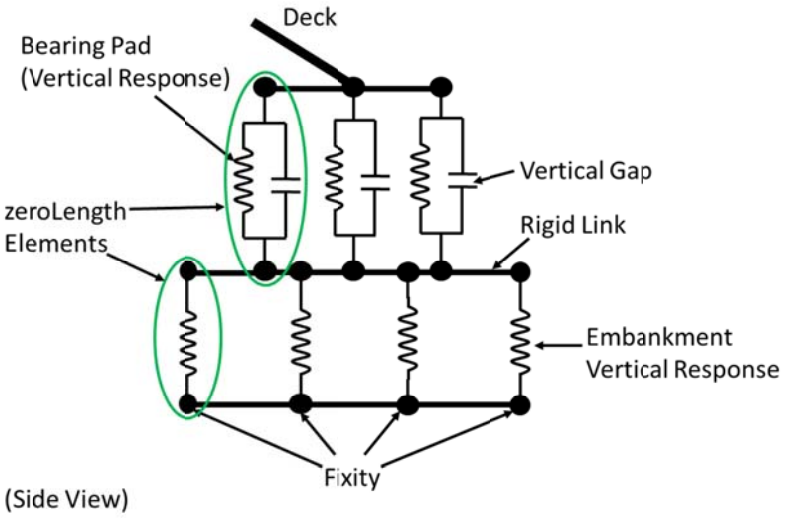
The vertical response of the abutment model includes the vertical stiffness of the bearing pads in series with the vertical stiffness of the trapezoidal. The detailed scheme of the vertical response is shown in Fig. 66a. The typical vertical response of a bearing pad is shown in Fig. 66b. And the typical overall behavior of the vertical response is illustrated in Fig. 66c.

A vertical gap (2-inch by default, which can be modified by the user) is employed for the vertical property of the bearing pads. The embankment stiffness per unit length of embankment was obtained from Zhang and Makris (2000) and modified using the critical length to obtain a lumped stiffness.

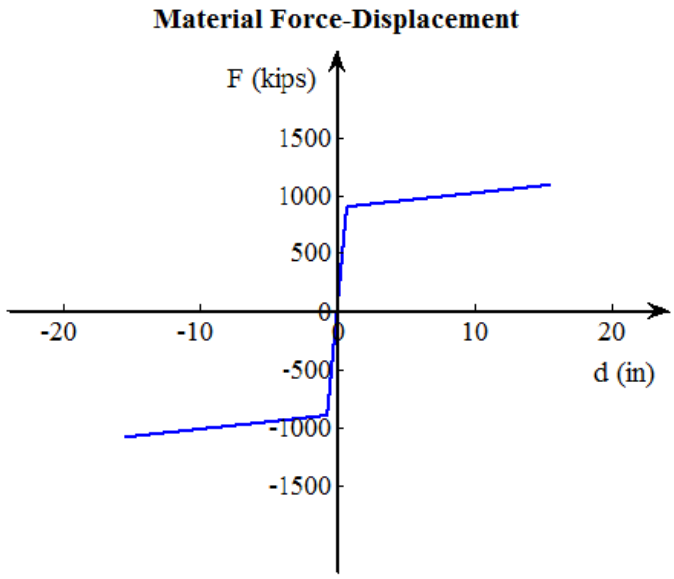
In the vertical direction, an elastic spring is defined at each end of the rigid link, with a stiffness corresponding to the vertical stiffness of the embankment soil mass. The embankment is assumed to have a trapezoidal shape and based on the effective length formulas from Zhang and Makris (2002), the vertical stiffness (K_v , unit: 1/m) can be calculated from (Zhang and Makris, 2002):

$$K_v = \frac{E_{sl} d_w}{z_0 \ln\left(\frac{z_0 + H}{z_0}\right)} L_c \tag{3}$$

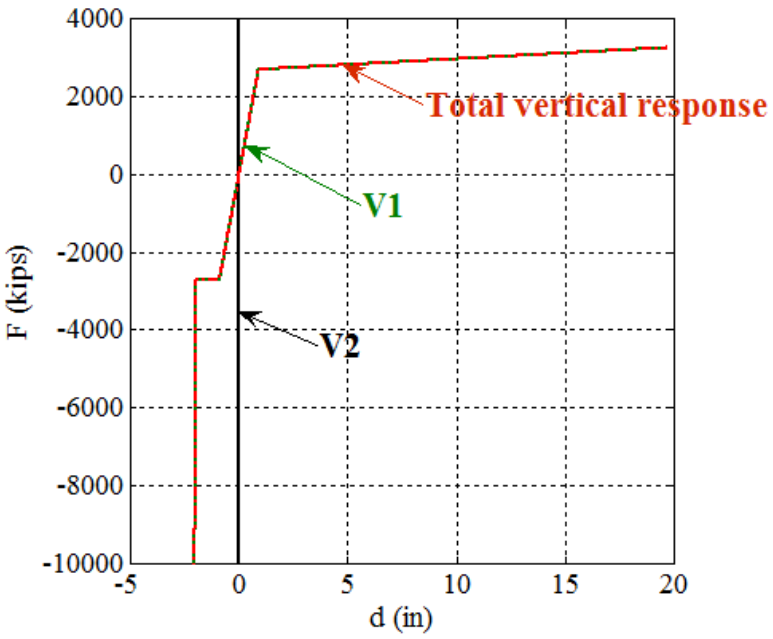
Where H is the embankment height, d_w is the deck width, $z_0 = 0.5 d_w S$, S is the embankment slope (parameter in window, see Fig. 20), $E_{sl} = 2.8G$, $G = \rho V_s^{\,2}$, ρ and V_s are the mass density and the shear wave velocity of the embankment soil, respectively.



a)



b)

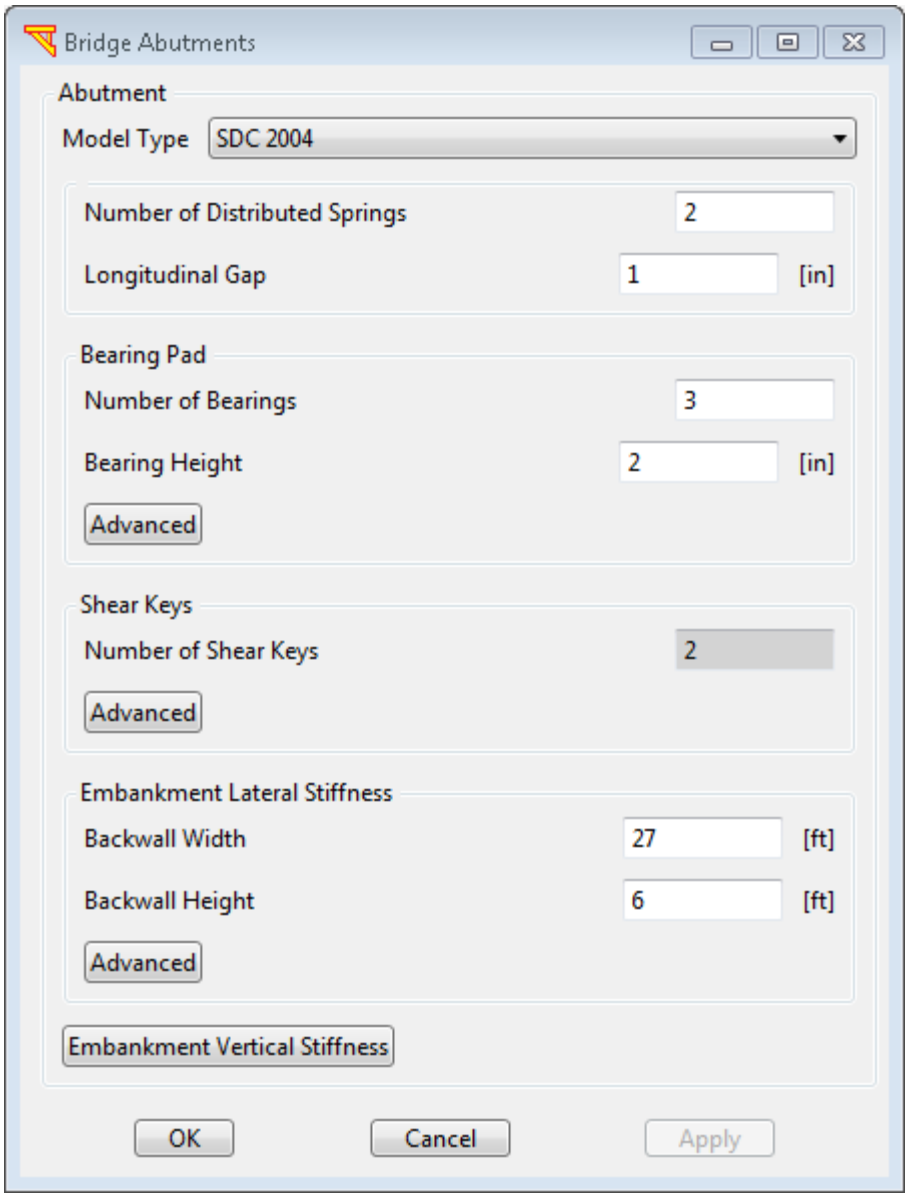


c)

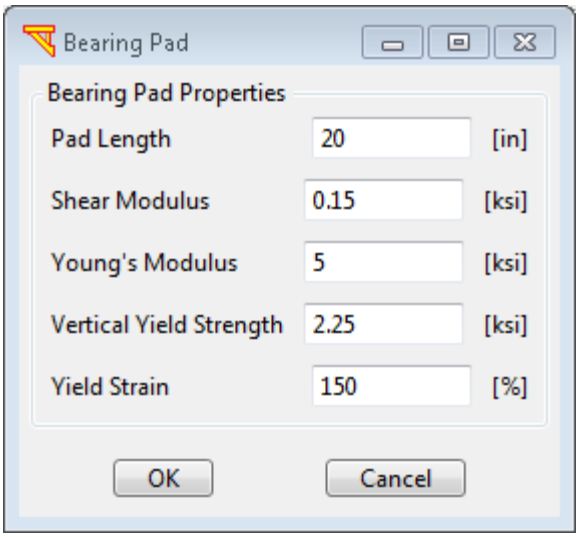
Fig. 66. Vertical response of the SDC 2004 Abutment Model: a) general scheme; b) vertical response of a bearing pad; c) total vertical response

4.3.4 Definition of the SDC 2004 Abutment Model

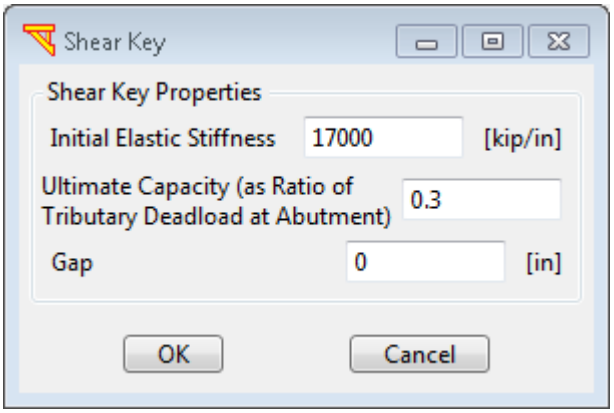
To define a SDC 2004 Abutment Model, please follow the steps shown in Fig. 67. To define a SDC 2004 Abutment Model, select **SDC 2004** for the abutment model type in Fig. 55. The resulting window is shown in Fig. 67a.


Bridge Abutments
Abutment
Model Type
SDC 2004
Number of Distributed Springs
2
Longitudinal Gap
1
[in]
Bearing Pad
Number of Bearings
3
Bearing Height
2
[in]
Advanced
Shear Keys
Number of Shear Keys
2
Advanced
Embankment Lateral Stiffness
Backwall Width
27
[ft]
Backwall Height
6
[ft]
Advanced
Embankment Vertical Stiffness
OK
Cancel
Apply

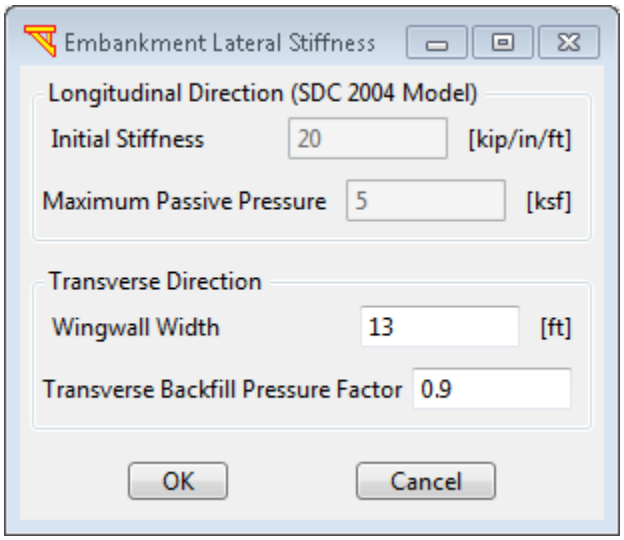
a)


Bearing Pad
Bearing Pad Properties
Pad Length
20
[in]
Shear Modulus
0.15
[ksi]
Young's Modulus
5
[ksi]
Vertical Yield Strength
2.25
[ksi]
Yield Strain
150
[%]
OK
Cancel

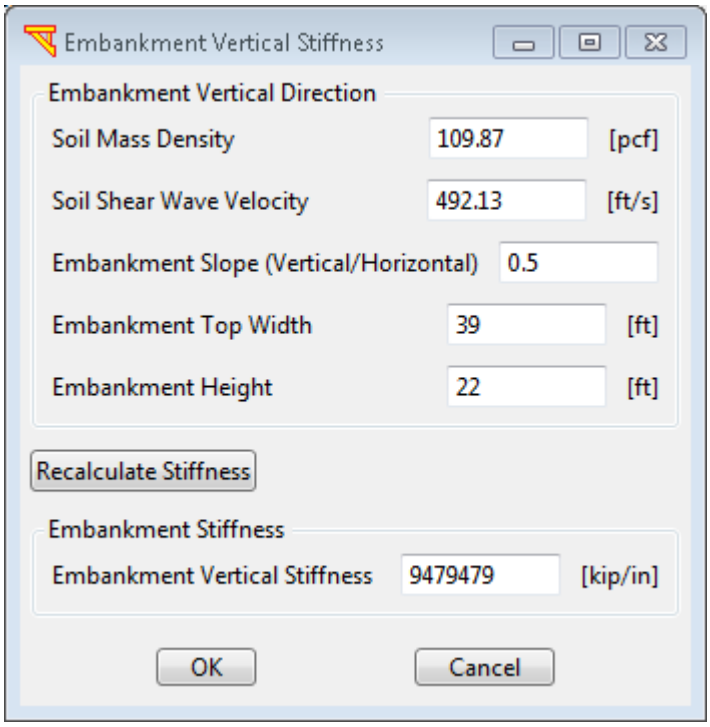
b)


Shear Key
Shear Key Properties
Initial Elastic Stiffness
17000
[kip/in]
Ultimate Capacity (as Ratio of Tributary Deadload at Abutment)
0.3
Gap
0
[in]
OK
Cancel

c)


Embankment Lateral Stiffness
Longitudinal Direction (SDC 2004 Model)
Initial Stiffness
20
[kip/in/ft]
Maximum Passive Pressure
5
[ksf]
Transverse Direction
Wingwall Width
13
[ft]
Transverse Backfill Pressure Factor
0.9
OK
Cancel

d)



e)

Fig. 67. Definition of the SDC 2004 Abutment Model: a) main parameters; b) bearing pad properties; c) shear key properties; d) SDC abutment properties; e) embankment properties

4.4 SDC 2010 Sand Abutment Model

This model is similar to the SDC 2004 abutment model, but employs the parameters of the most recent SDC 2010 for a sand backfill Embankment (Fig. 68). To define a SDC 2010 Sand Abutment Model, select **SDC 2010 Sand** for the abutment model type in Fig. 55. Table 5 shows the initial stiffness and the maximum passive pressure employed for the SDC 2010 Sand Abutment Model, compared to other similar abutment models including SDC 2004, SDC 2010 Clay, EPP-Gap and HFD Models).

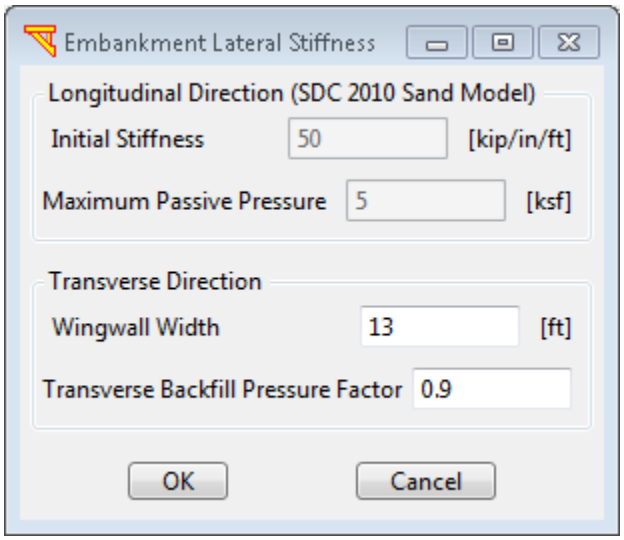


Fig. 68. Backfill horizontal properties for the SDC 2010 Sand Abutment Model

Table 5. SDC Abutment Properties

Abutment Model	Initial Stiffness (kip/in/ft)	Maximum Passive Pressure (ksf)
SDC 2004	20	5
SDC 2010 Sand	50	5
SDC 2010 Clay	25	5
EPP-Gap	User-defined	User-defined
HFD Model	*50 (sand); 25 (clay)	5.5

*Denotes average soil stiffness K50.

4.5 SDC 2010 Clay Abutment Model

This model is similar to the SDC 2004 abutment model, but employs the parameters of the most recent SDC 2010 for a Clay backfill Embankment (Fig. 69). To define a SDC 2010 Clay Abutment Model, select **SDC 2010 Clay** for the abutment model type in Fig. 67a. Table 5 shows the initial stiffness and the maximum passive pressure employed for the SDC 2010 Clay Abutment Model, compared to other similar abutment models.

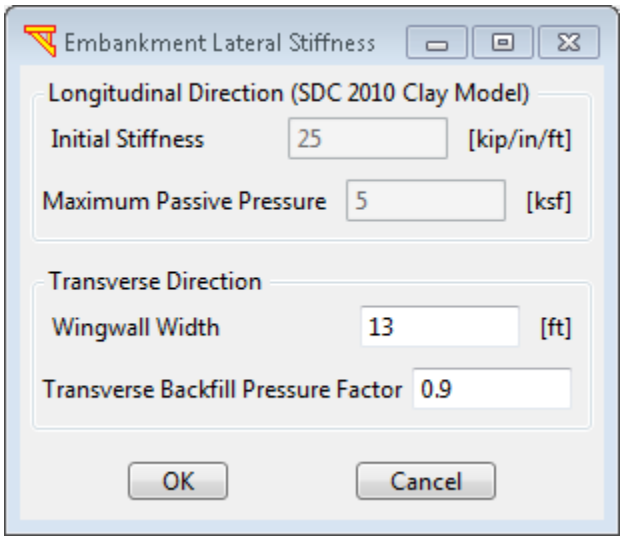


Fig. 69. Backfill horizontal properties of the SDC 2010 Clay Abutment Model

4.6 ElasticPP-Gap Model

This model is similar to the SDC 2004 Abutment Model, but employs user defined parameters for the stiffness and maximum resistance (Fig. 70). To define an EPP-Gap Abutment Model, select **EPP-Gap** for the abutment model type in Fig. 67.

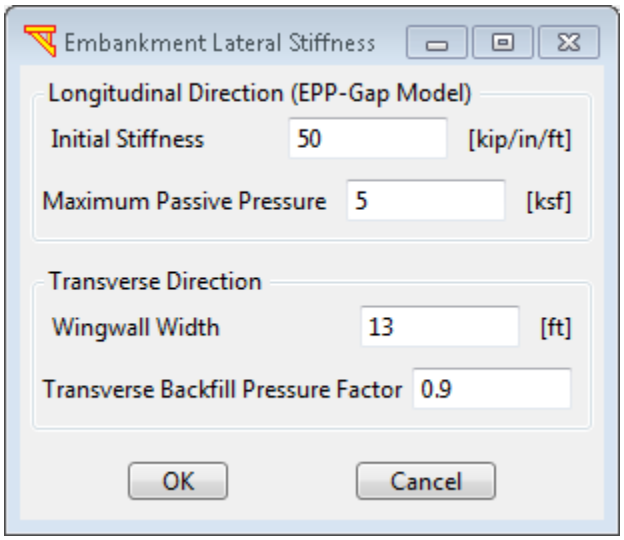


Fig. 70. Backfill horizontal properties of the EPP-Gap Abutment Model

4.7 HFD Model

As suggested by Shamsabadi et al. (2007, 2010), a Hyperbolic Force-Displacement (HFD) relationship is employed to represent abutment resistance to bridge displacement in the longitudinal direction (Fig. 71).

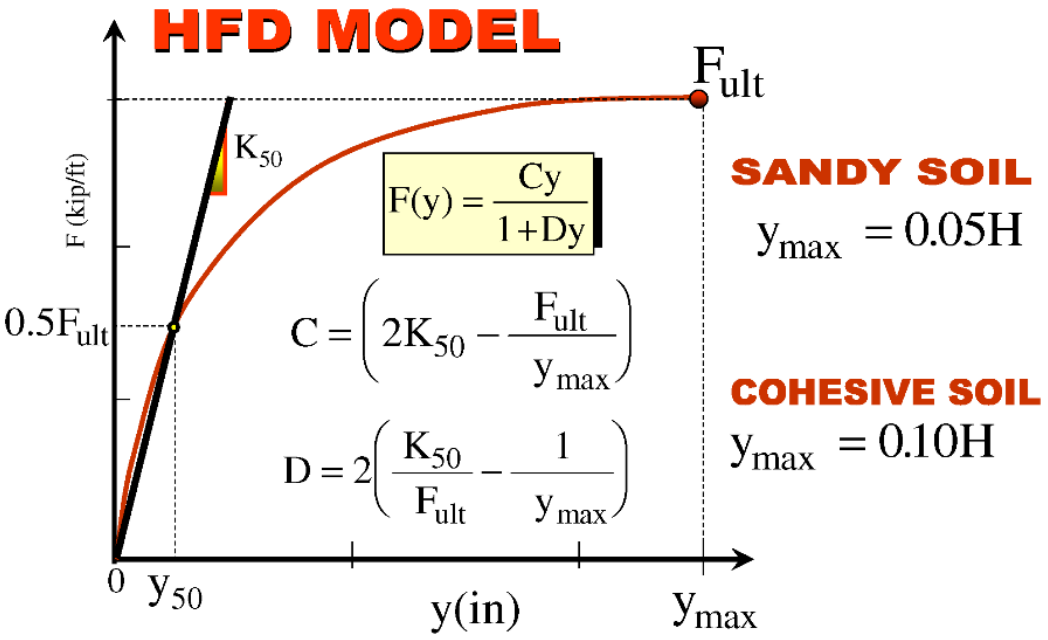
$$F(y) = \frac{F_{ult}(2K_{50}y_{max} - F_{ult})y}{F_{ult}y_{max} + 2(K_{50}y_{max} - F_{ult})y}$$

Where F is the resisting force, y is the longitudinal displacement, F_{ult} is the ultimate passive resistance and K_{50} is the secant stiffness at $F_{ult}/2$.

$$F(y) = \frac{\left(2K_{50} - \frac{F_{ult}}{y_{max}}\right)y}{1 + 2\left(\frac{K_{50}}{F_{ult}} - \frac{1}{y_{max}}\right)y}$$

In this HFD model, resistance appears after a user-specified gap is traversed, and the bridge thereafter gradually mobilizes the abutment's passive earth pressure strength. Herein, this strength is specified according to Shamsabadi et al. (2007, 2010) at 5.5 ksf (for a nominal 5.5 ft bridge deck height), with full resistance occurring at a passive lateral displacement of 3.6 in (the sand structural backfill scenario). Similarly, abutment resistance to the transverse bridge displacement is derived from the longitudinal hyperbolic force-displacement relationship according to the procedure outlined in Aviram et al. (2008).

To define a HFD abutment model, select **HFD Model** for the abutment model type in Fig. 67. Click **Advanced** in **Embankment Lateral Stiffness** box (Fig. 67) to define the backfill horizontal properties (Fig. 71c). Parameters of the backfill soil are defined based on soil types (sand, clay, or User-defined) and the overall abutment stiffness/ or maximum passive pressure resist are calculated using the SDC equations.



a)

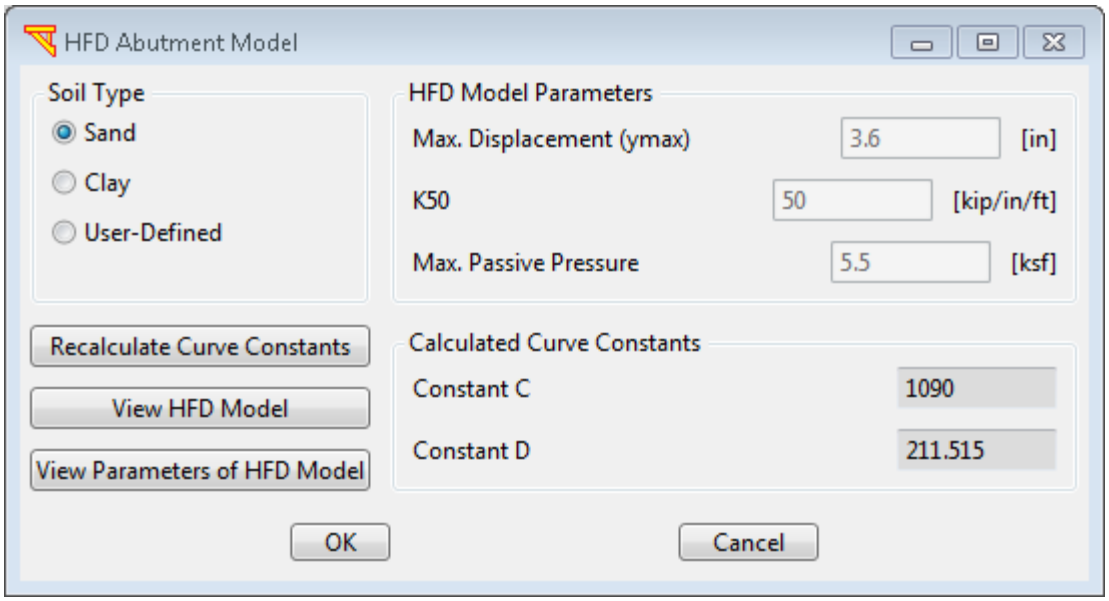
Table 4. Suggested HFD Parameters for Abutment Backfills

Abutment backfill type	Pressure kPa (ksf)	Average soil stiffness kN/cm/m (K/in/ft)	Maximum displacement y_{max}/H
Granular[a]	265 (5.5)	290 (50)	0.05
Cohesive[a]	265 (5.5)	145 (25)	0.10

Note: Abutment backwall height = 1.67 m (5.5 ft).

[a]Compacted to at least 95% relative compaction per ASTM D-1557.

b)



c)

Fig. 71. Definition of the HFD Abutment Model: a) HFD abutment model; and b) HFD parameters for abutment backfills suggested by Shamsabadi et al. (2007); and c) backfill properties of the HFD Model

5 Column Responses & Bridge Resonance

MSBridge provides features to view column lateral responses, abutment responses and bridge natural periods (Fig. 7 and Fig. 72).

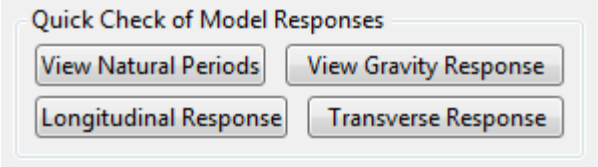


Fig. 72. Buttons to view column & abutment responses and bridge resonance

5.1 Bridge Natural Periods

Click **View Natural Periods (**Fig. 72**)** to view the natural periods and frequencies of the bridge (Fig. 73). A mode shape analysis is conducted in this case.

The user can copy and paste the values to their favorite text editor such as MS Excel (in Fig. 73, right-click and then click **Select All** (ctrl a) to highlight, and then right-click and then click **Copy** (ctrl c) to copy to the clipboard).

5.2 Column Gravity Response

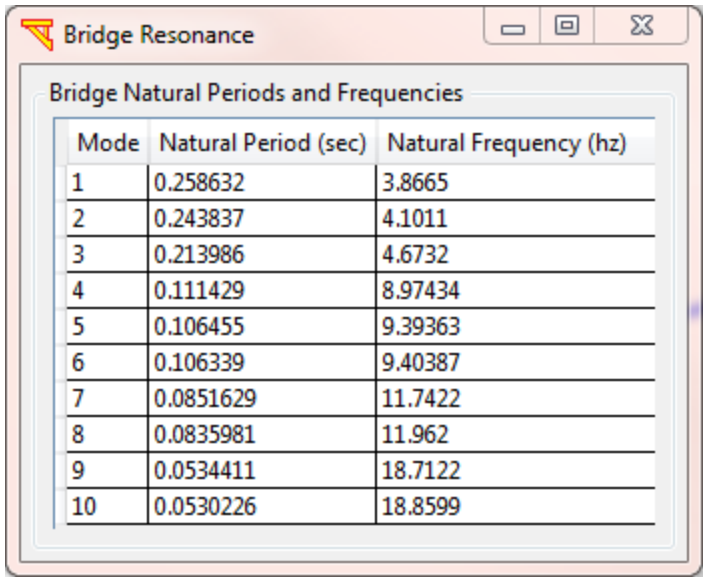
Click **View Gravity Response (**Fig. 72**)** to view the column internal forces and bending moments after application of own weight (Fig. 74).

5.3 Column & Abutment Longitudinal Responses

Click **Longitudinal Response (**Fig. 72**)** to view the column longitudinal responses (Fig. 75) and the abutment longitudinal responses (Fig. 76). A pushover up to 5% of drift ratio in the longitudinal direction is conducted in this case.

5.4 Column & Abutment Transverse Responses

Click **Transverse Response (**Fig. 72**)** to view the column transverse responses (Fig. 77) and the abutment transverse responses (Fig. 78). A pushover up to 4% of drift ratio in the transverse direction is conducted in this case.



Mode	Natural Period (sec)	Natural Frequency (hz)
1	0.258632	3.8665
2	0.243837	4.1011
3	0.213986	4.6732
4	0.111429	8.97434
5	0.106455	9.39363
6	0.106339	9.40387
7	0.0851629	11.7422
8	0.0835981	11.962
9	0.0534411	18.7122
10	0.0530226	18.8599

Fig. 73. Natural periods and frequencies of bridge

Bridge Gravity Response

Bridge Column Forces and Bending Moments Under Deadload

Column	Location	Axis Force [kip]	Longitudinal Shear [kip]	Transverse Shear [kip]	Longitudinal Moment [kip]	Transverse Moment [kip]
Column 1 of Bent 1	Column Top	-385.009	7.13743	-46.6324	-105.993	-684.25
Column 1 of Bent 1	Column Bottom	-417.093	7.13743	-46.6324	50.9154	341.013
Column 2 of Bent 1	Column Top	-385.009	7.13743	46.6324	-105.993	684.25
Column 2 of Bent 1	Column Bottom	-417.093	7.13743	46.6324	50.9154	-341.013
Column 1 of Bent 2	Column Top	-326.47	-1.68785e-10	-39.3141	6.84948e-09	-576.912
Column 1 of Bent 2	Column Bottom	-358.554	-1.24809e-09	-39.3141	-1.30778e-08	287.449
Column 2 of Bent 2	Column Top	-326.47	-1.68797e-10	39.3141	6.84955e-09	576.912
Column 2 of Bent 2	Column Bottom	-358.554	-1.24809e-09	39.3141	-1.30778e-08	-287.449
Column 1 of Bent 3	Column Top	-385.009	-7.13743	-46.6324	105.993	-684.25
Column 1 of Bent 3	Column Bottom	-417.093	-7.13743	-46.6324	-50.9154	341.013
Column 2 of Bent 3	Column Top	-385.009	-7.13743	46.6324	105.993	684.25
Column 2 of Bent 3	Column Bottom	-417.093	-7.13743	46.6324	-50.9154	-341.013

Fig. 74. Column internal forces and bending moments after application of own weight

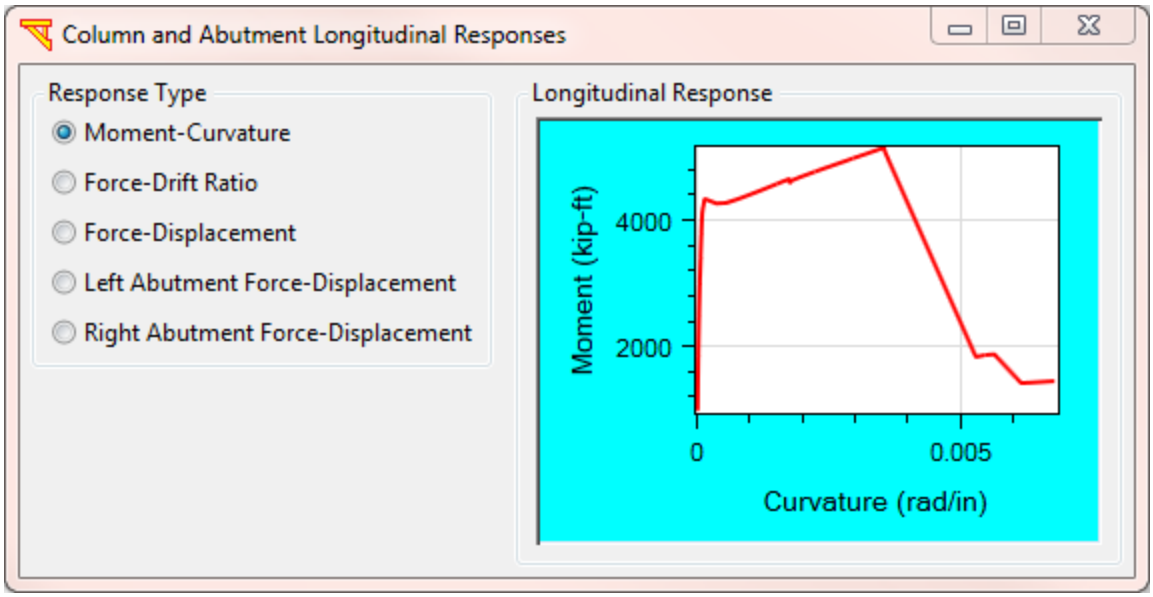


Fig. 75. Column longitudinal responses

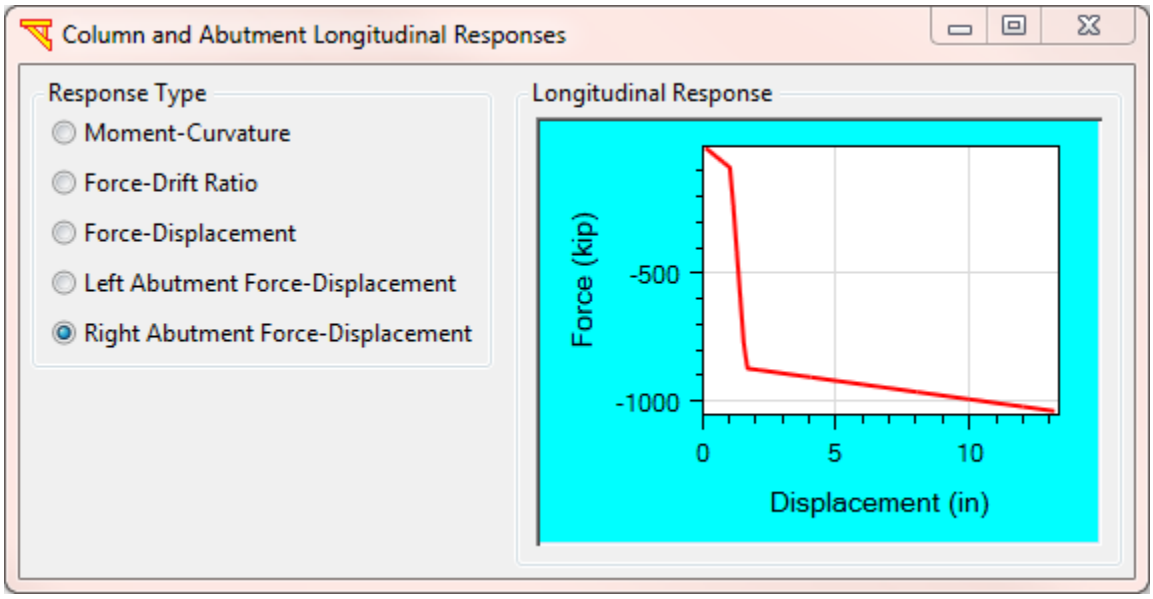


Fig. 76. Abutment longitudinal responses

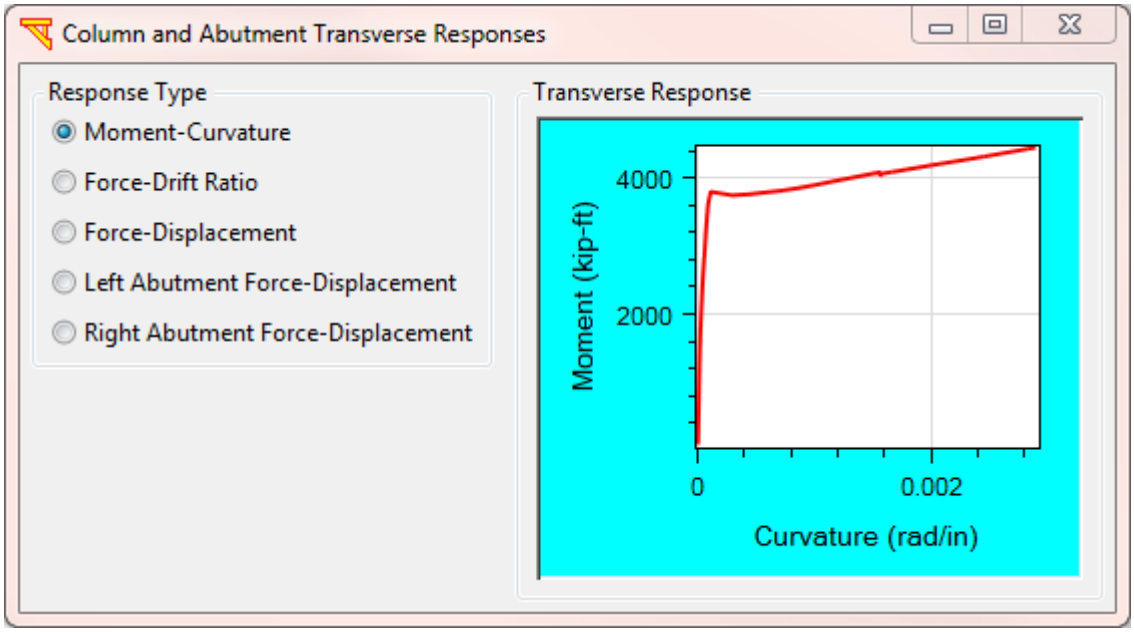


Fig. 77. Column transverse responses

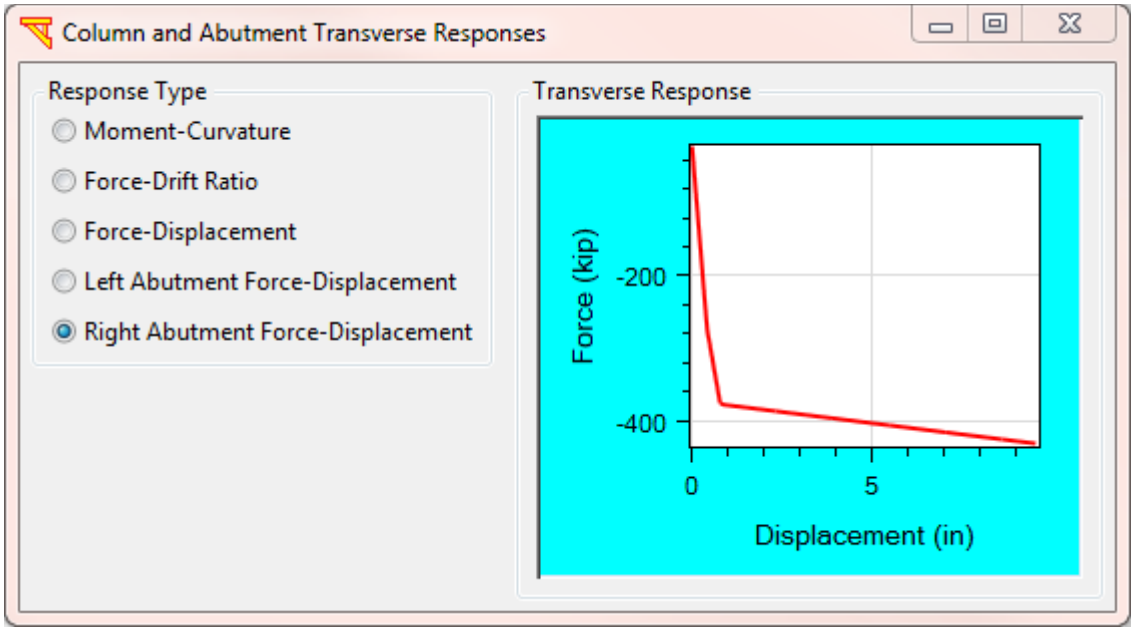


Fig. 78. Abutment transverse responses

6 Pushover & Eigenvalue Analyses

To conduct a pushover analysis, a load pattern must be defined. As shown in Fig. 79, first, choose **Pushover** in the Analysis Options, and then click **Change Pattern**. The load pattern window is shown in Fig. 80.

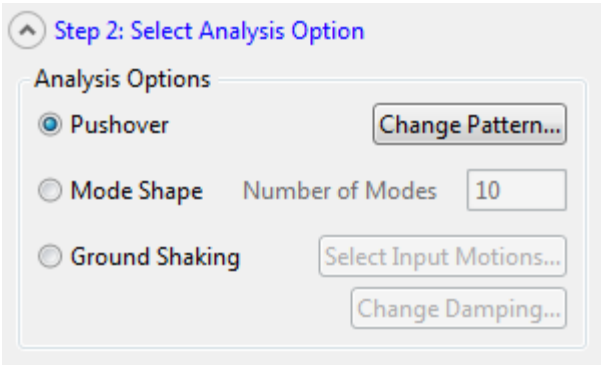


Fig. 79. Pushover analysis option

6.1 Monotonic Pushover

The pushover options include **Monotonic Pushover**, **Cyclic Pushover**, and **U-Push** (pushover by a user-defined loading pattern).

Two methods of pushover are available (Fig. 80): force-based and displacement-based. If **Force-Based Method** is chosen, please enter the parameters of force increment (per step): **Longitudinal (X) Force**, **Transverse (Y) Force**, **Vertical (Z) Force**, **Moment @ X**, **Moment @ Y**, and **Moment @ Z**.

If **Displacement-Based Method** is chosen, please enter the displacement increment parameters (per step): **Longitudinal Displacement**, **Transverse Displacement**, **Vertical Displacement**, **Rotation around X**, **Rotation around Y**, and **Rotation around Z**.

The pushover load/displacement linearly increases with step in a monotonic pushover mode.

The pushover load/displacement is applied at the bridge deck center or the deck location at a bent.

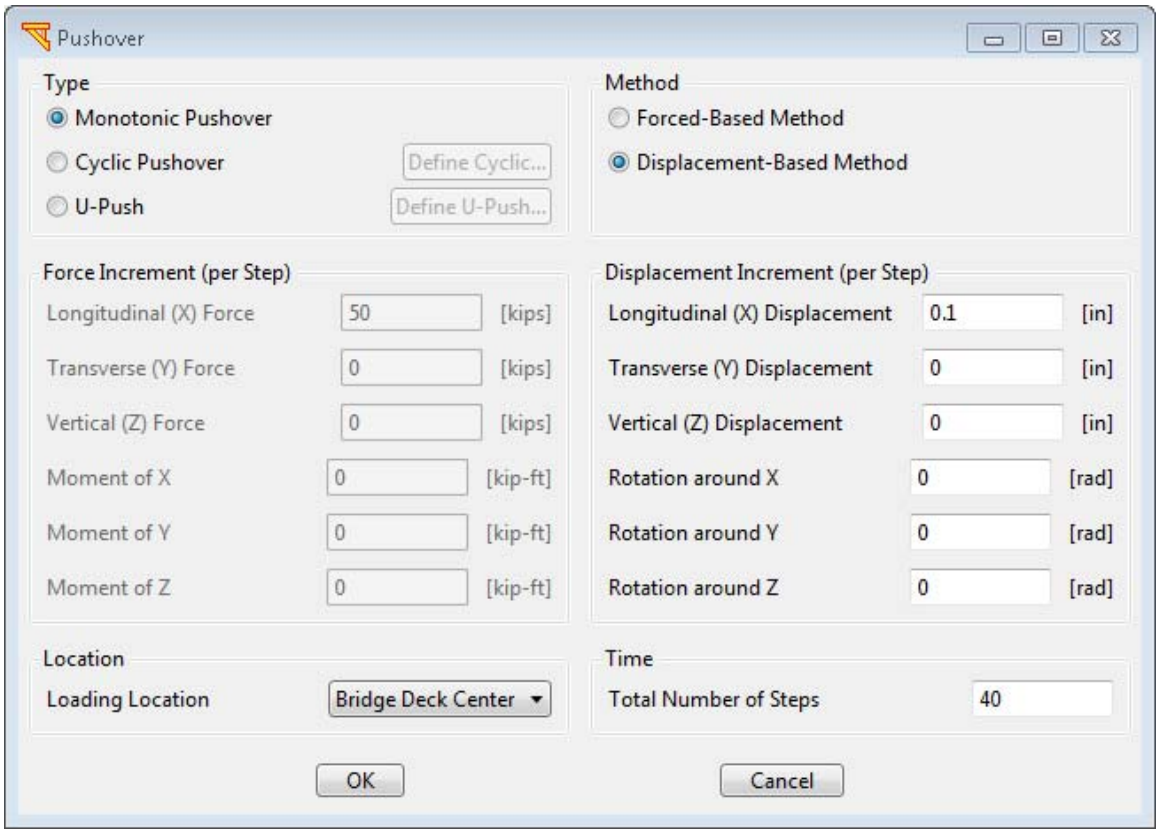


Fig. 80. Load pattern for monotonic pushover analysis

6.2 Cyclic Pushover

To conduct a Cyclic Pushover, click **Cyclic Pushover** in Fig. 80 and then define Number of **Steps for the First Cycle** and **Step Increment per Cycle (**Fig. 81).

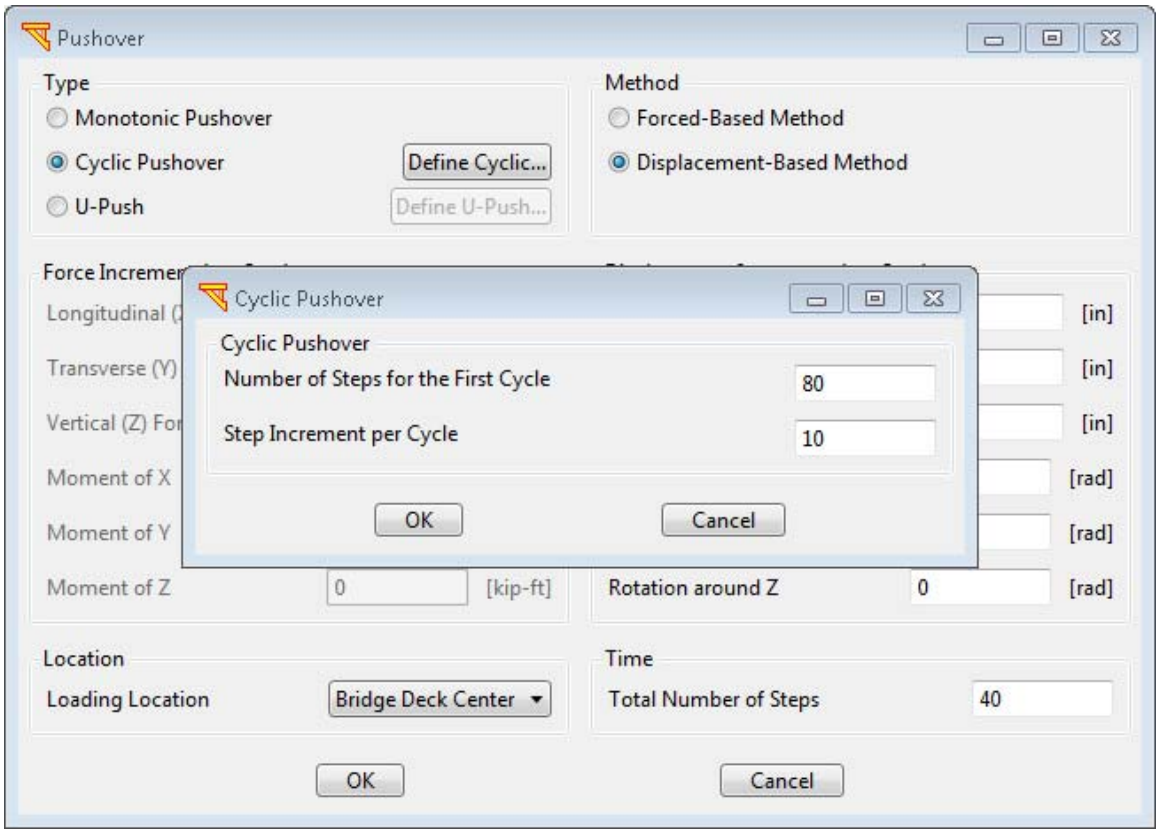


Fig. 81. Load pattern for cyclic pushover analysis

6.3 User-Defined Pushover (U-Push)

Click **U-Push** and then click **Define U-Push** to enter your own load pattern (U-Push). In this case, the displacement or force parameters entered in Fig. 82 are used as the maximum values. The U-Push data entered are used as the factors (of the maximum displacement or the maximum force entered).

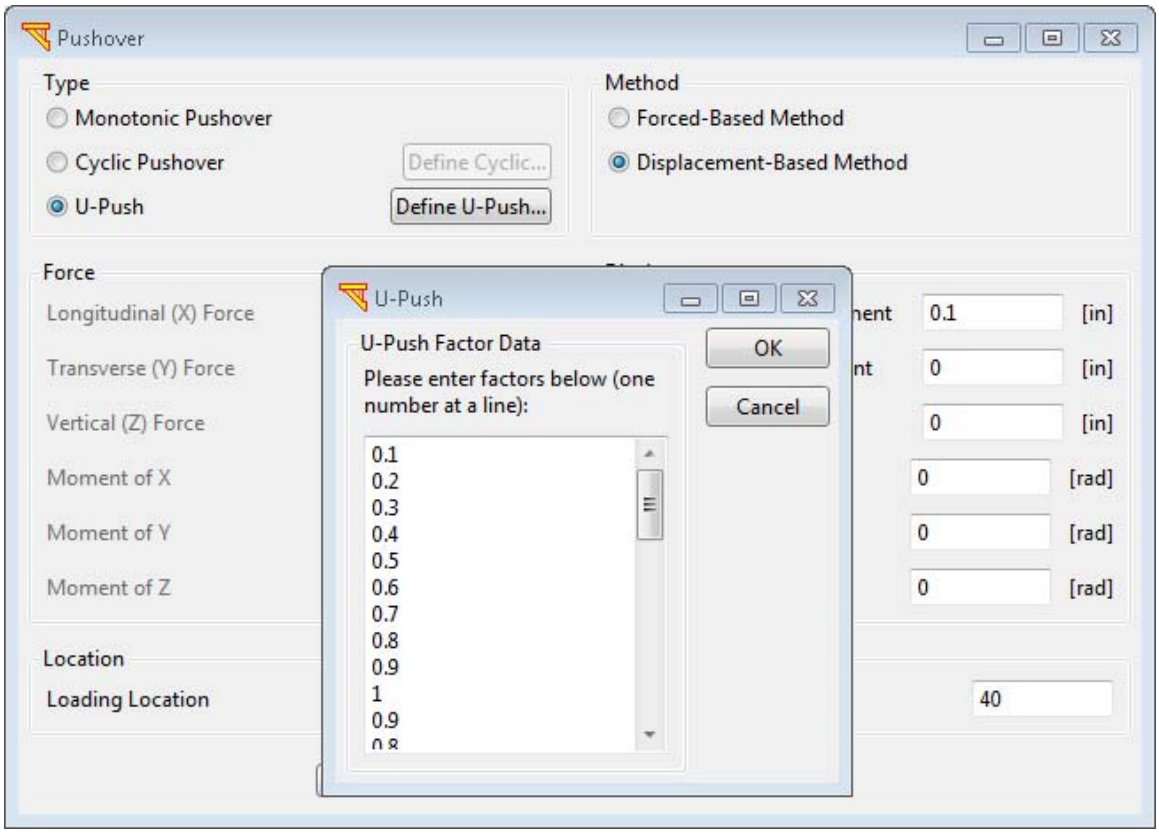


Fig. 82. User-defined pushover (U-Push)

6.4 Output for Pushover Analysis

Output windows for a pushover analysis include:

i) Response time histories and profiles for column (and pile shaft under grade)
ii) Response relationships (force-displacement as well as moment-curvature) for column (and pile shaft under grade)
iii) Abutment response time histories
iv) Deformed mesh, contour fill, plastic hinges, and animations.

6.4.1 Column Response Profiles

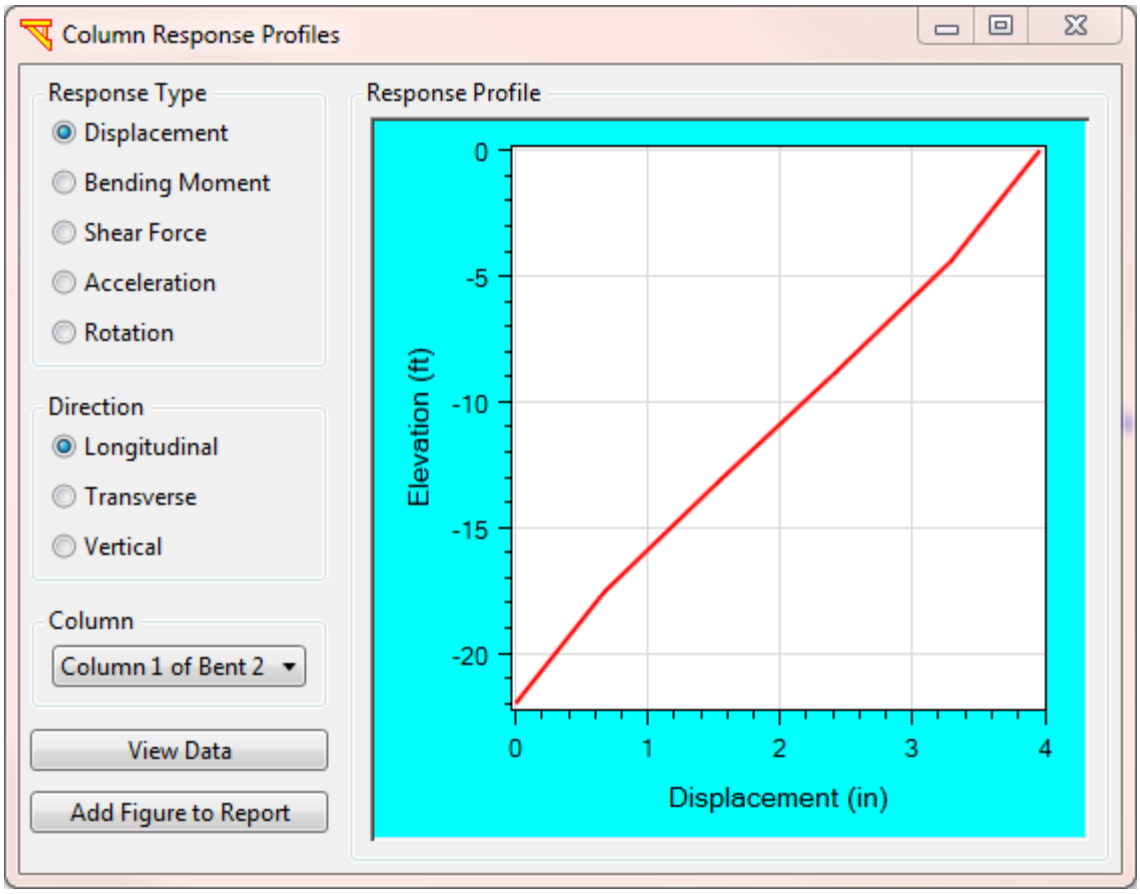


Fig. 83. Column response profiles

6.4.2 Column Response Time Histories

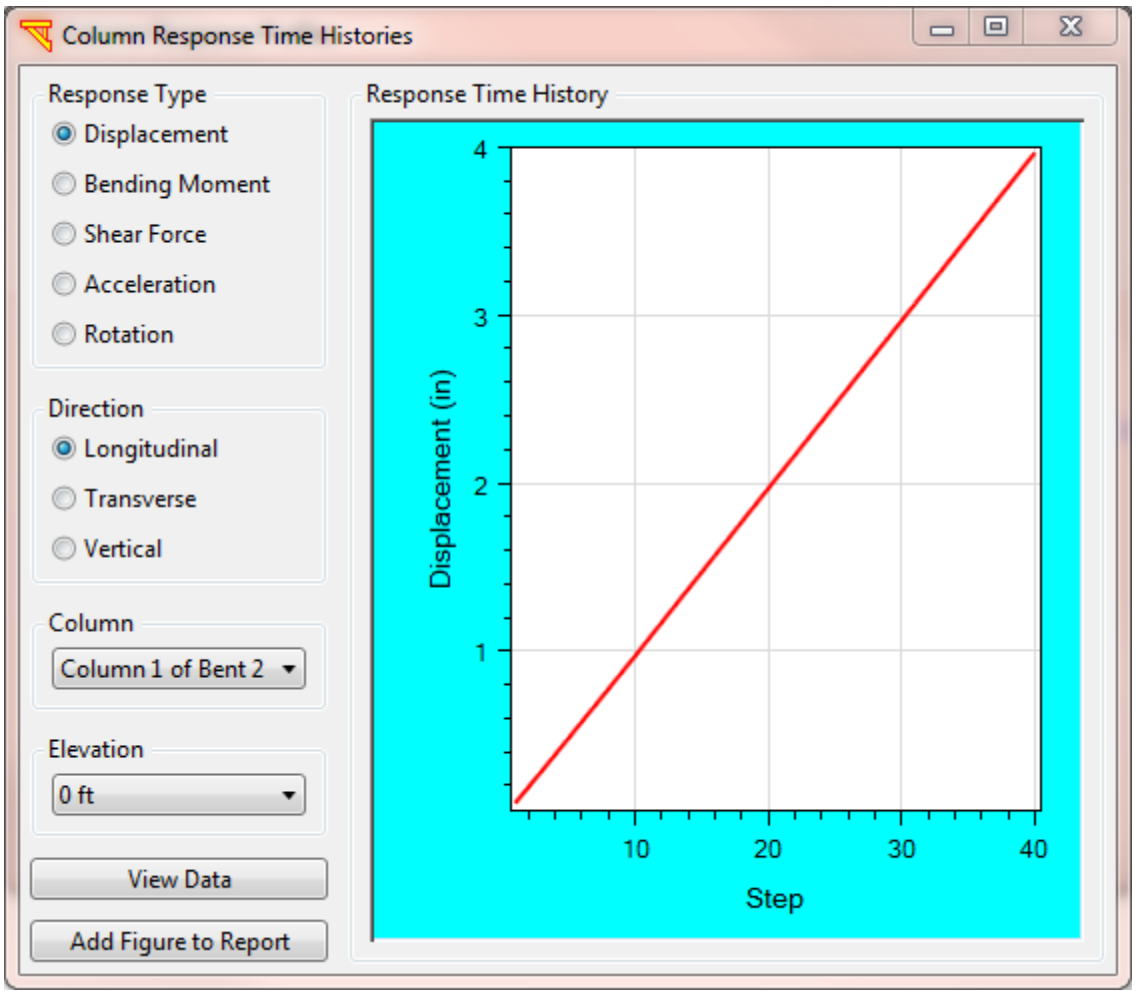


Fig. 84. Column response time histories

6.4.3 Column Response Relationships

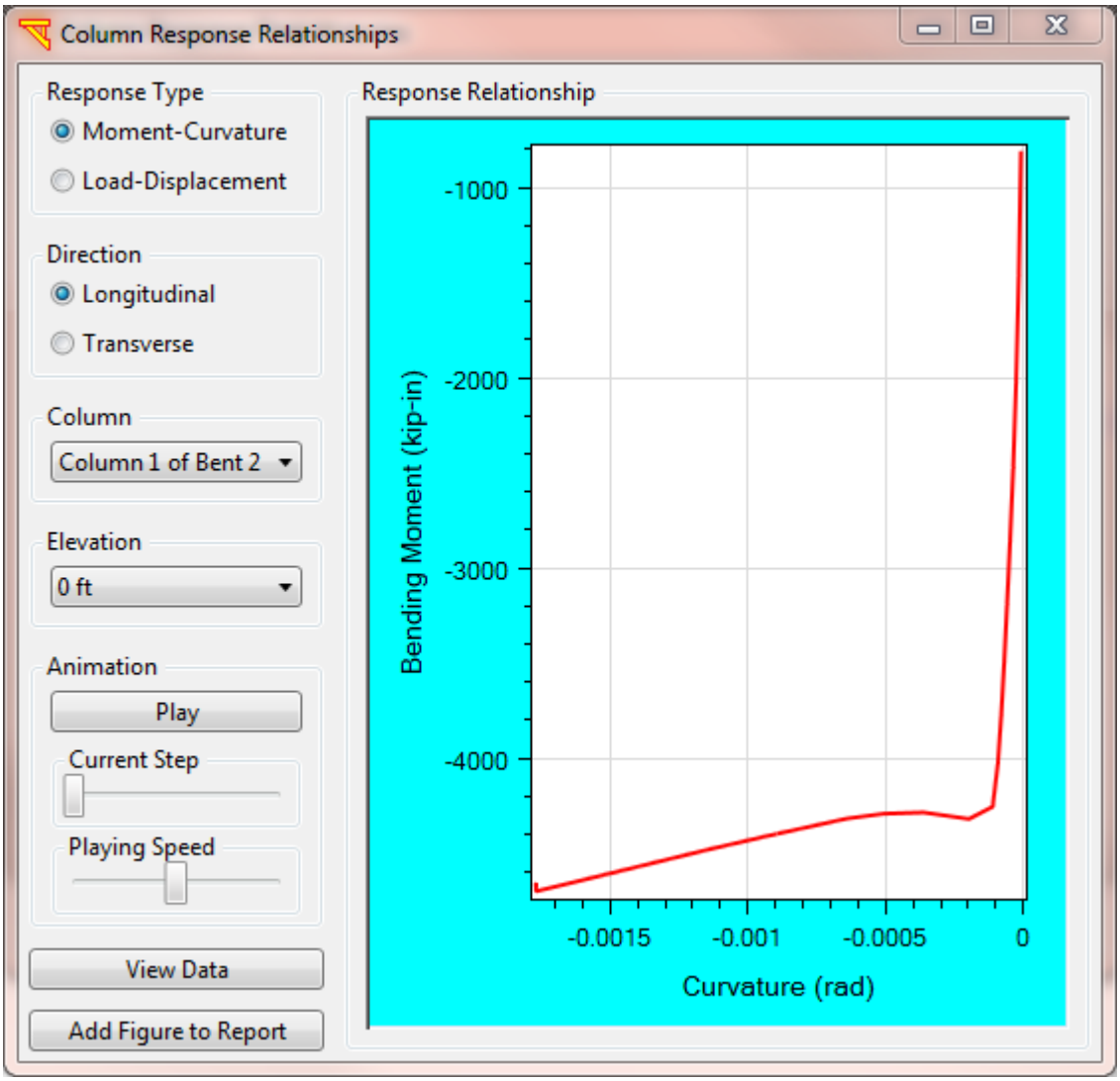


Fig. 85. Response relationships for column

6.4.4 Abutment Force-Displacement and Response Time Histories

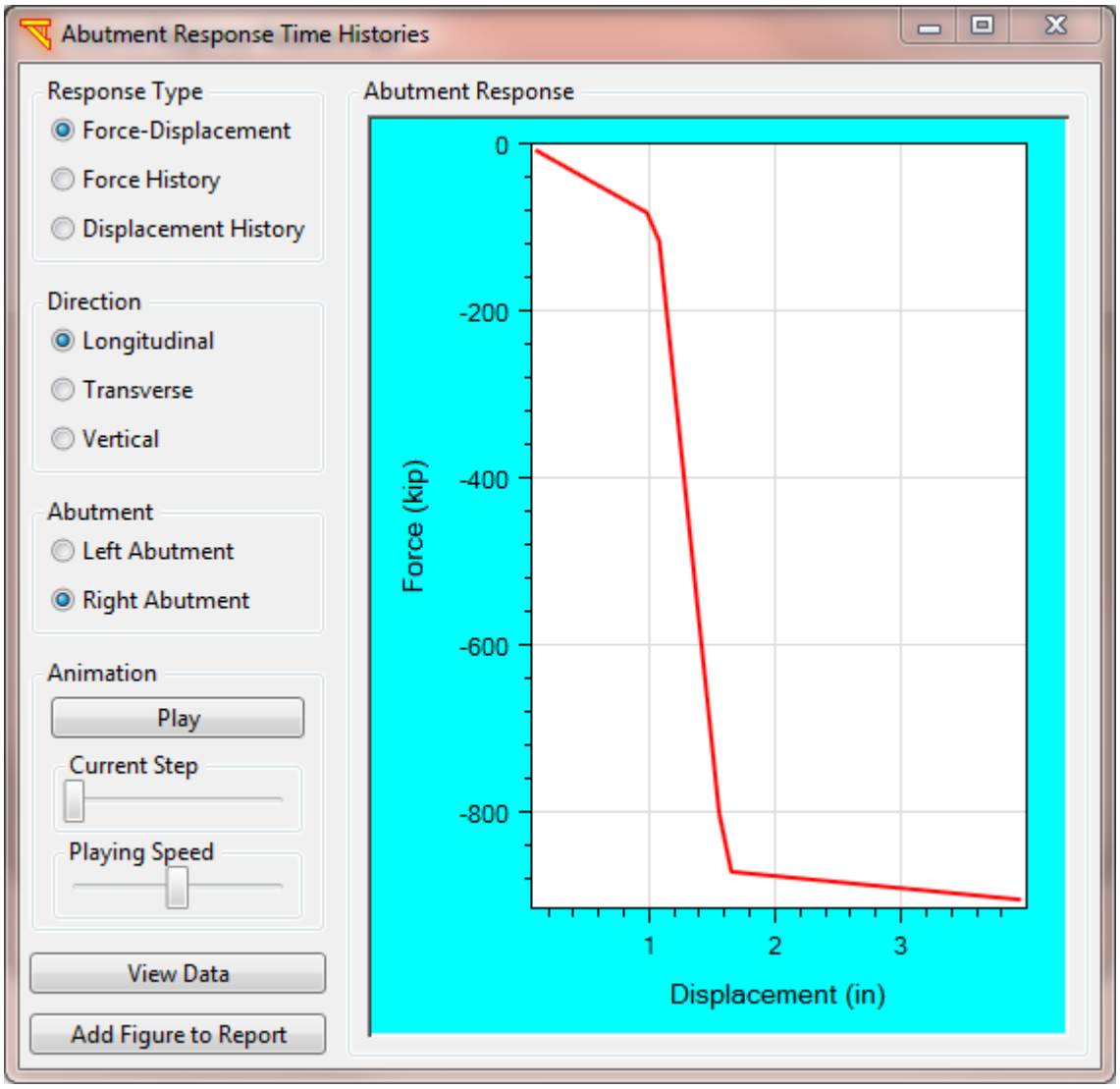


Fig. 86. Abutment response time histories

6.4.5 Deformed Mesh

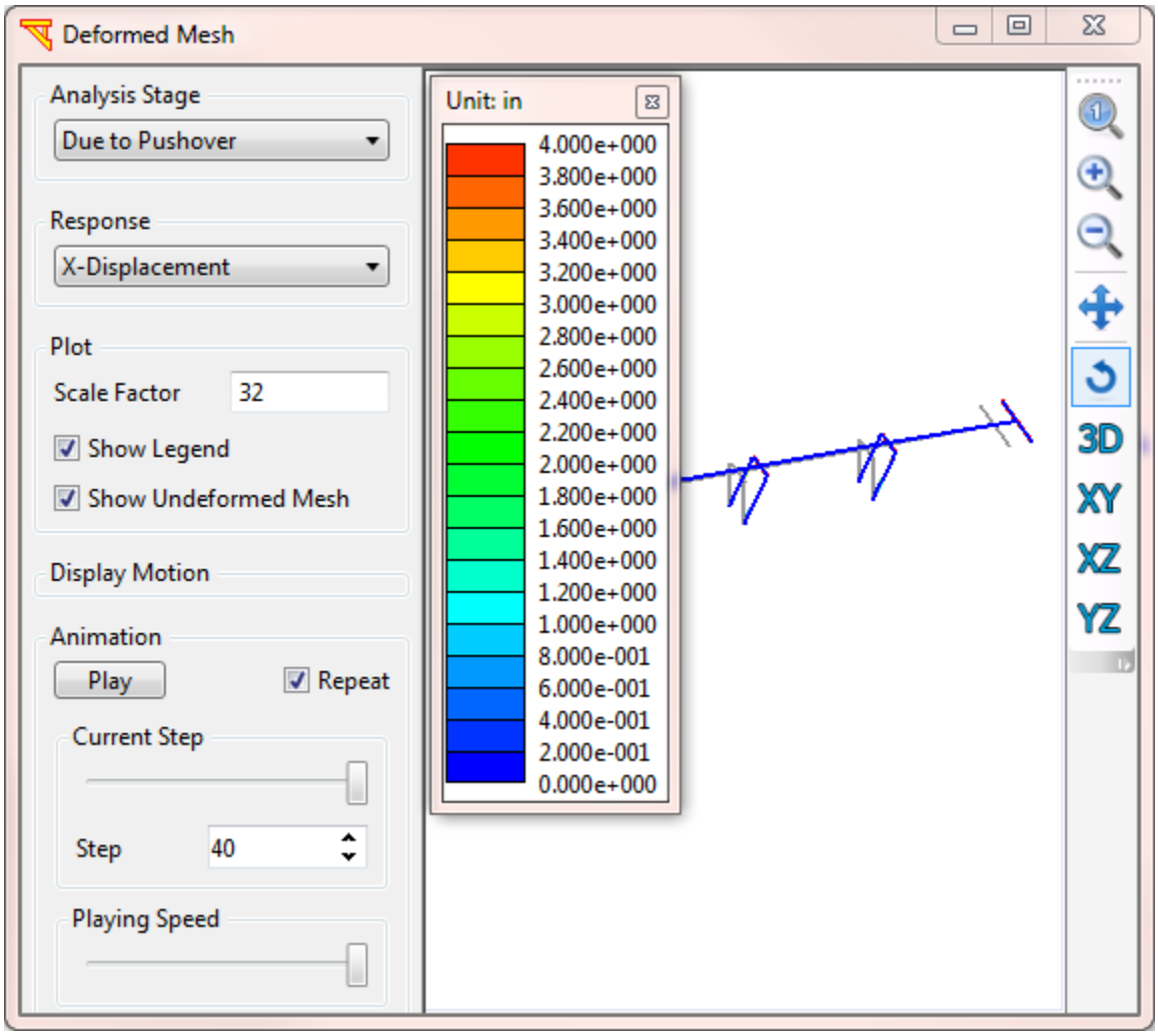


Fig. 87. Deformed mesh and contour fill

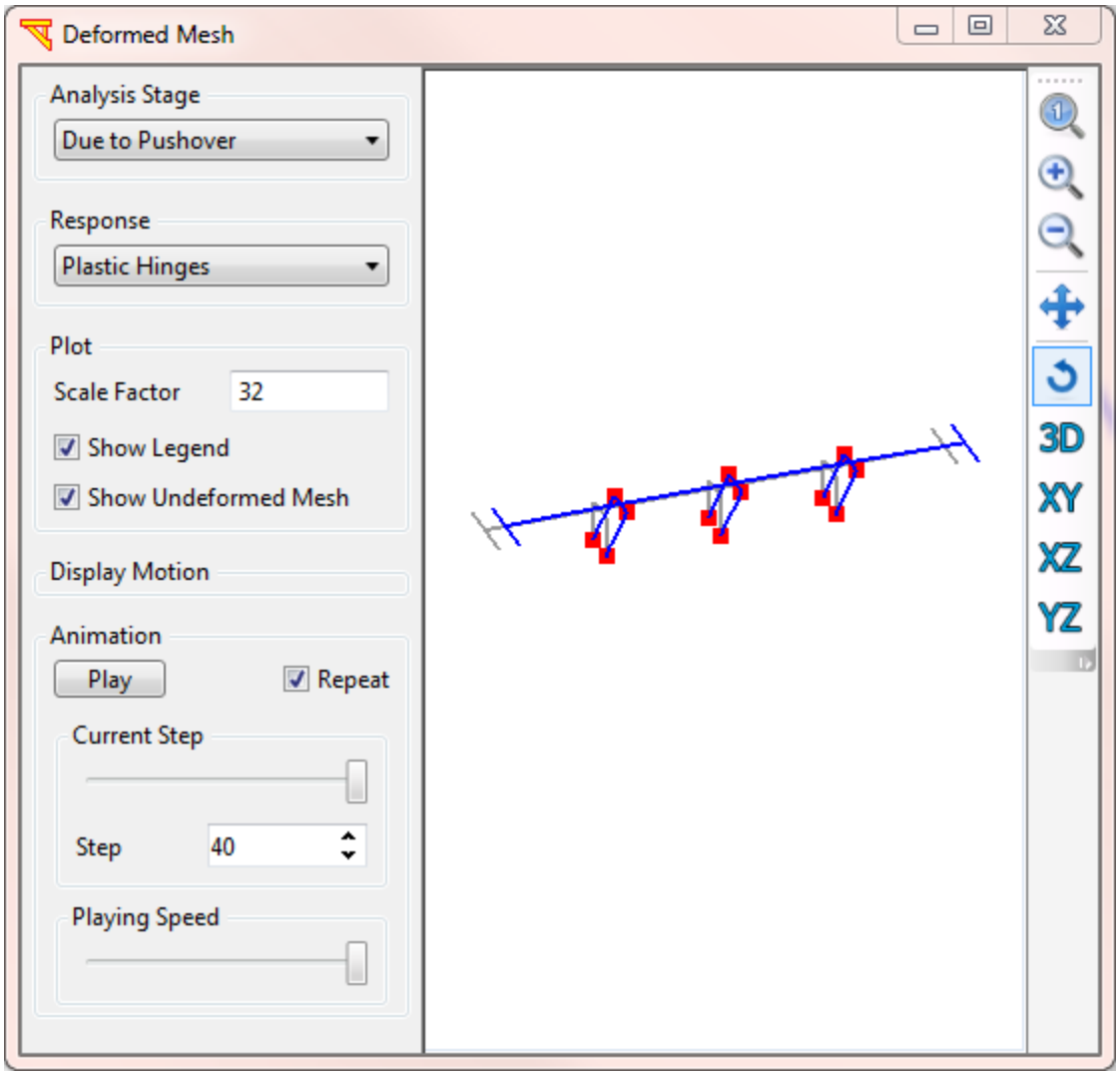


Fig. 88. Visualization of Plastic hinges (red dots represent plastic hinges developed)

6.5 Eigenvalue Analysis

To conduct an Eigenvalue analysis, please follow the steps shown in Fig. 89 and then click **Save Model & Run** Analysis. Fig. 90 shows the output window for an Eigenvalue analysis, which can be accessed by clicking menu Display (Fig. 3) and then choosing Deformed Mesh. To switch between modes, move the slider or click the spin button to cycle through them.

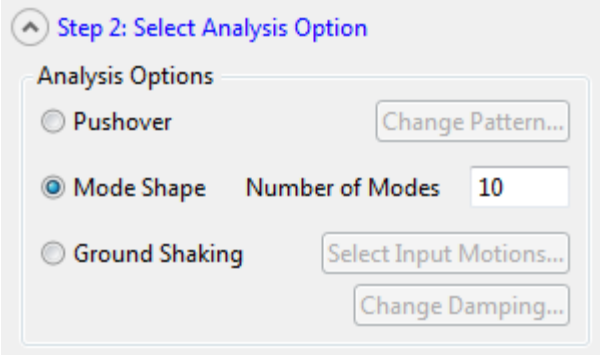
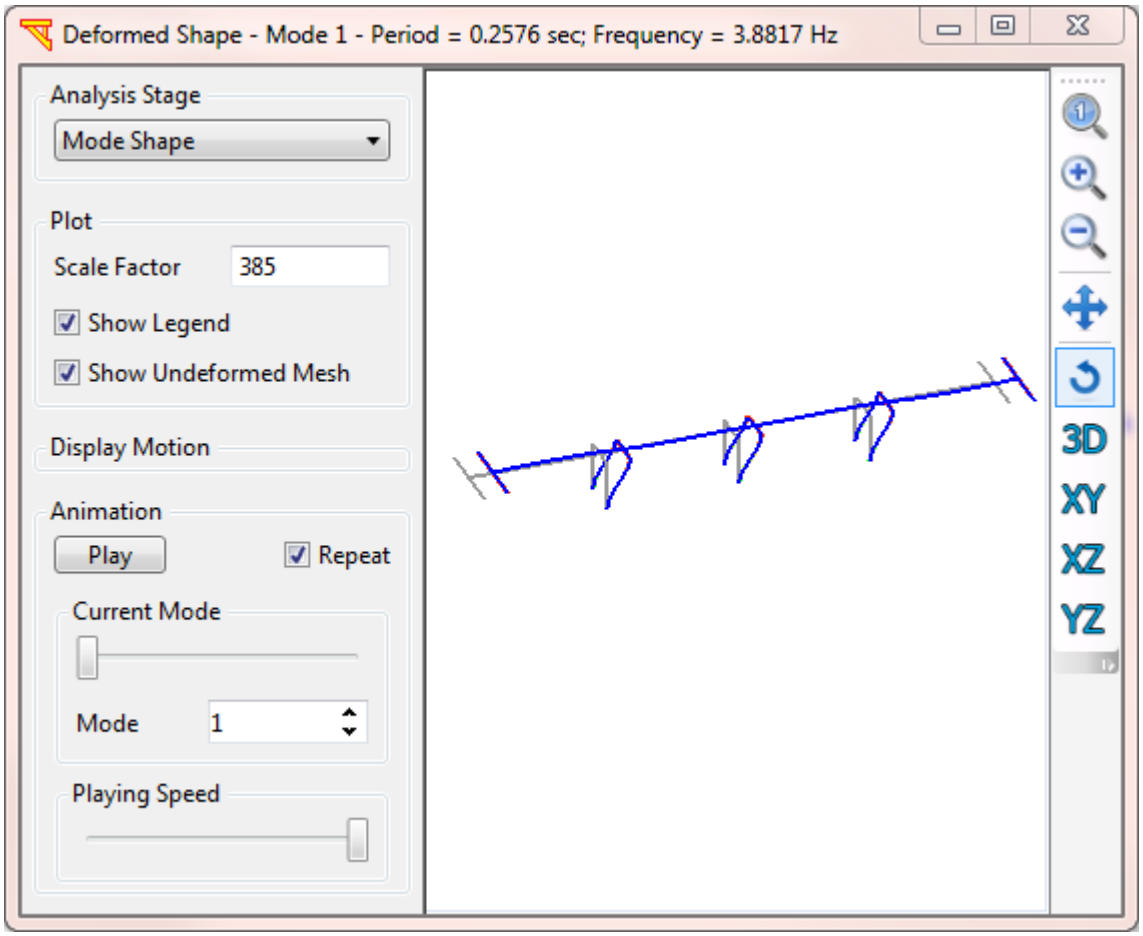
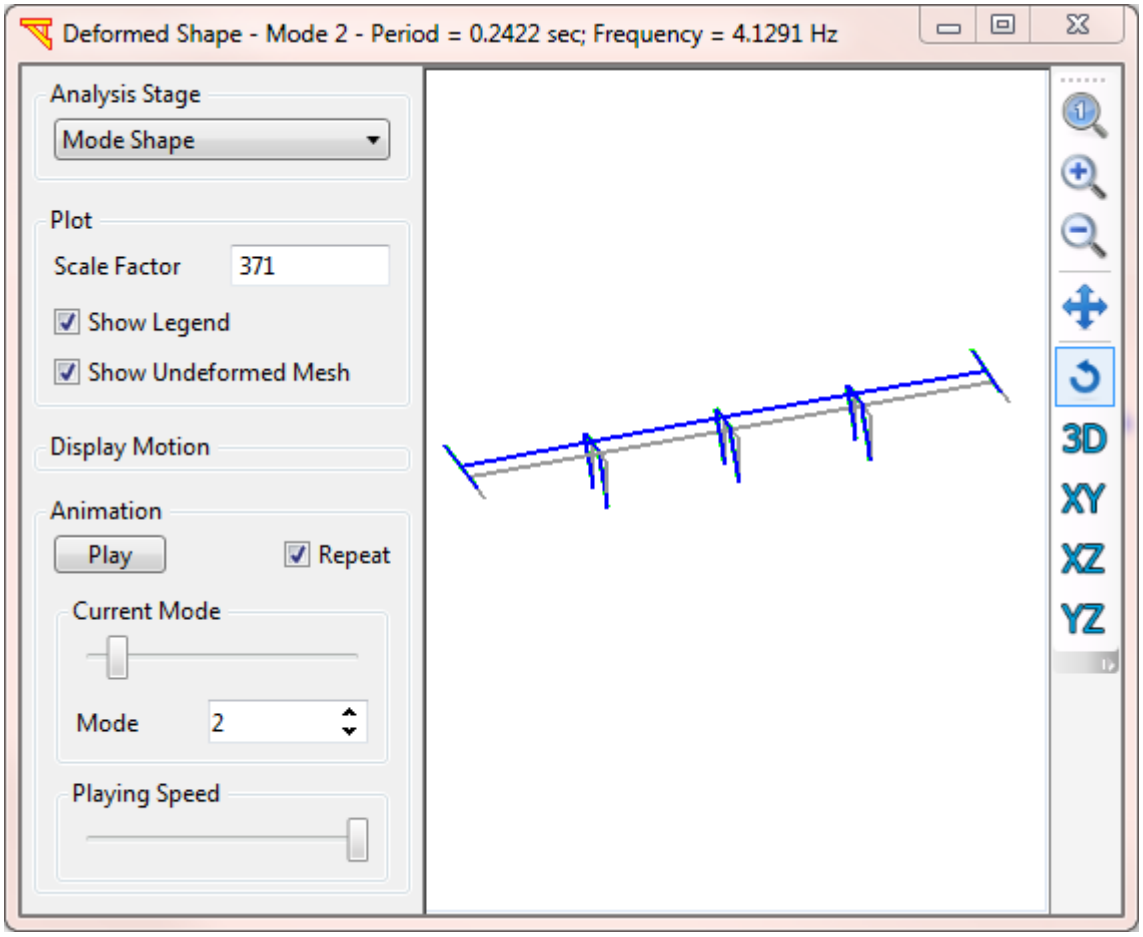


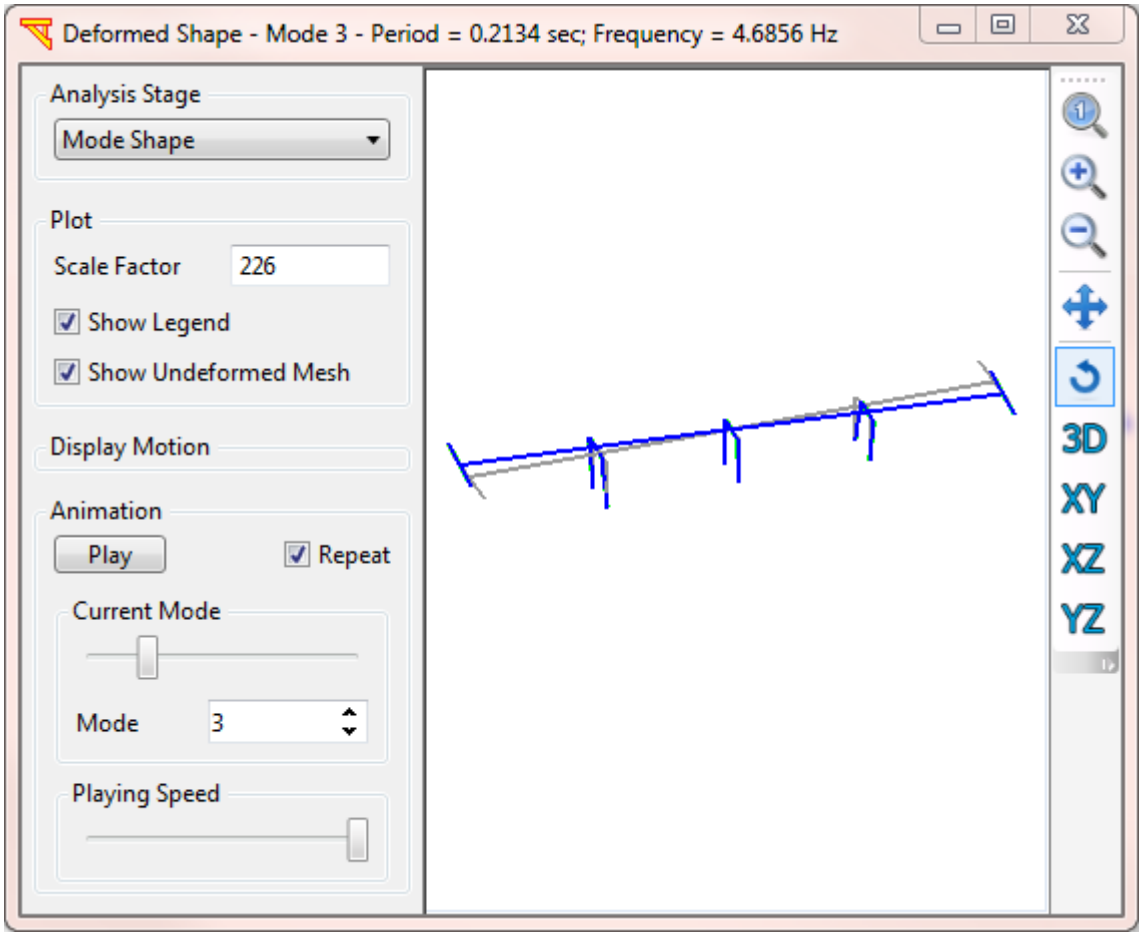
Fig. 89. Steps to perform an Eigenvalue analysis



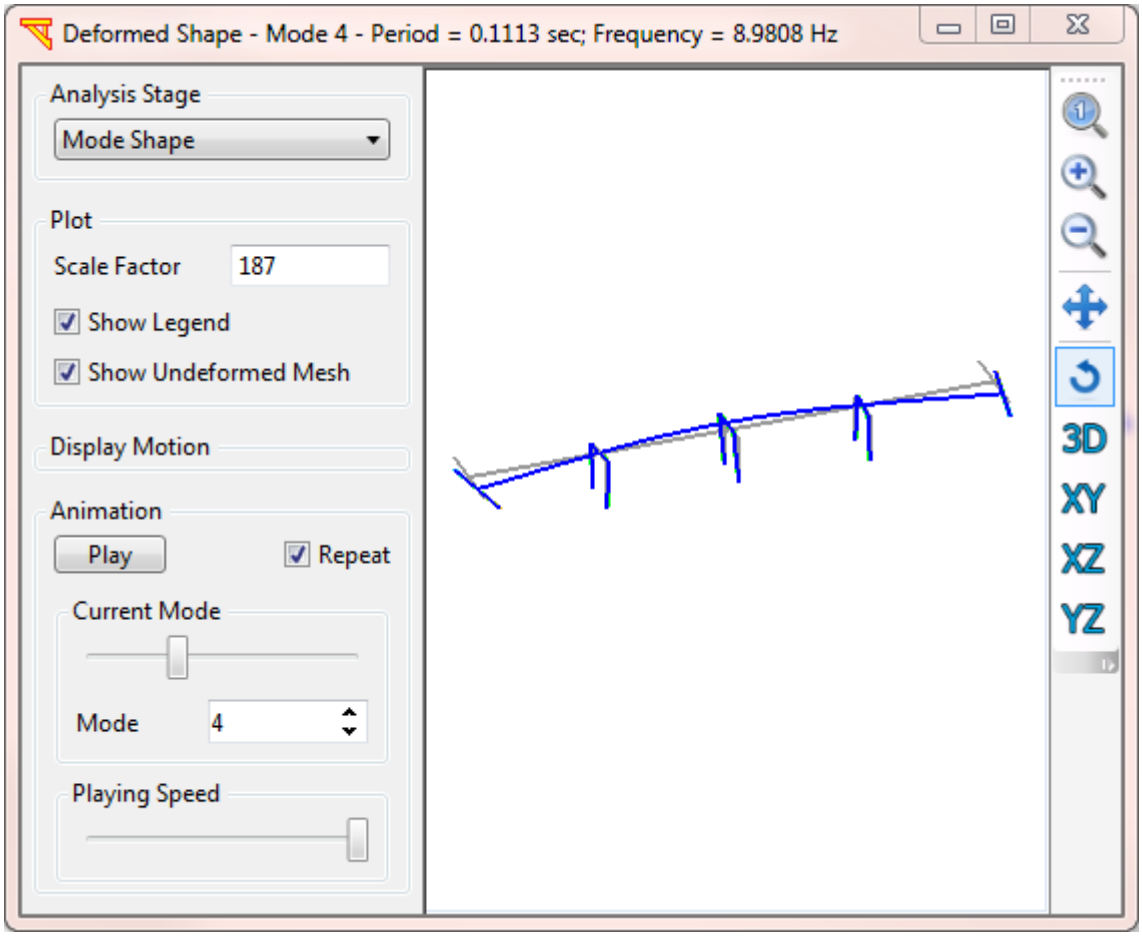
a)


Deformed Shape - Mode 2 - Period = 0.2422 sec; Frequency = 4.1291 Hz
Analysis Stage
Mode Shape
Plot
Scale Factor
371
Show Legend
Show Undeformed Mesh
Display Motion
Animation
Play
Repeat
Current Mode
Mode
2
Playing Speed
3D
XY
XZ
YZ

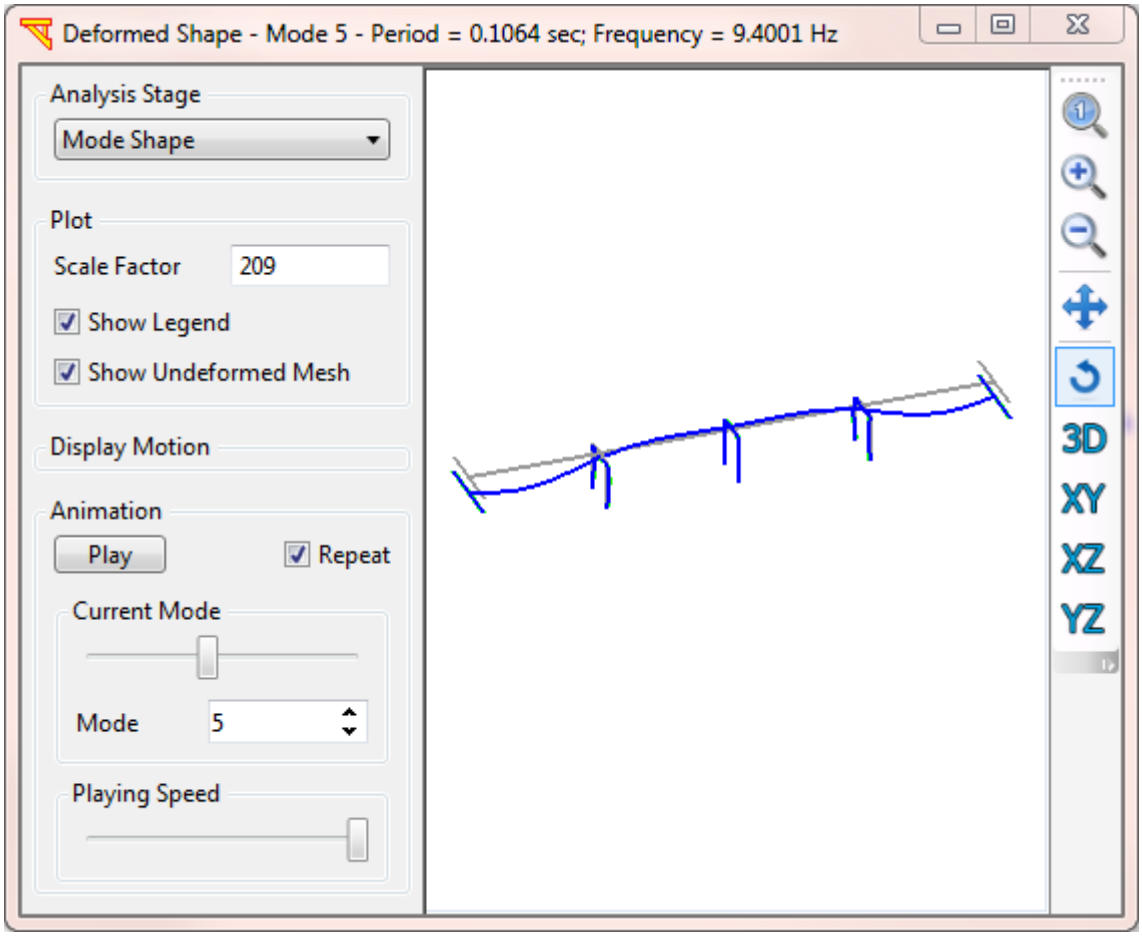
b)


Deformed Shape - Mode 3 - Period = 0.2134 sec; Frequency = 4.6856 Hz
Analysis Stage
Mode Shape
Plot
Scale Factor
226
Show Legend
Show Undeformed Mesh
Display Motion
Animation
Play
Repeat
Current Mode
Mode
3
Playing Speed
3D
XY
XZ
YZ

c)


Deformed Shape - Mode 4 - Period = 0.1113 sec; Frequency = 8.9808 Hz
Analysis Stage
Mode Shape
Plot
Scale Factor
187
Show Legend
Show Undeformed Mesh
Display Motion
Animation
Play
Repeat
Current Mode
Mode
4
Playing Speed
3D
XY
XZ
YZ

d)



e)

Fig. 90. Sample output for an Eigenvalue analysis for the default bridge model: a) first mode; b) second mode; c) third mode; d) fourth mode; and e) fifth mode

7 Ground Shaking

To conduct a single earthquake analysis or a multiple earthquake analyses, the "Ground Shaking" option under Analysis Options (Fig. 2 and Fig. 91) is used. For that purpose, the input earthquake excitation(s) must be specified. If only one earthquake record is selected out of a specified ensemble (suite) of input motions, then a conventional single earthquake analysis will be performed.

7.1 Definition/specification of input motion ensemble (suite)

7.1.1 Available Ground Motions

A set of 20 motions provided by CalTrans are available as the default input motion package. The above ground motion data sets were resampled to a sampling frequency of 50 Hz (regardless of whether initial sampling frequency was 100 or 200 Hz) due to the computational demands of running full ground-structure analyses for an ensemble of motions. Standard interpolation methods were used to resample the time domain signals (so that the signal shape is preserved). The resampled records were then baselined to remove any permanent velocity and displacement offsets. Baselining was accomplished using a third order polynomial fitted to the displacement record.

In addition, four sets of input motions are also available (can be downloaded from the website: http://www.soilquake.net/msbridge):

Motion Set 1: These 100 motions were obtained directly from the PEER NGA database and all files have been re-sampled to a time step of 0.02 seconds. This PBEE motion ensemble (Medina and Krawinkler 2004) obtained from the PEER NGA database (http://peer.berkeley.edu/nga/) consists of 100 3D input ground motions. Each motion is composed of 3 perpendicular acceleration time history components (2 lateral and one vertical). These motions were selected through earlier efforts (Gupta and Krawinkler, 2000; Mackie et al., 2007) to be representative of seismicity in typical regions of California. The moment magnitudes (Mw) of these motions range from 5.8-7.2 (distances from 0-60 km). The engineering characteristics of each motion and of the ensemble overall may be viewed directly within **MSBridge**. The provided ground motions are based on earlier PEER research (Mackie and Stojadinovic 2005).

Motion Set 2: These motions (160 in total) were developed by Dr. Kevin Mackie from the 80 motions of Set1 (excluding the 20 motions of Set1 in the bin NEAR), to account for site classification.

Motion Set 3: These motions (80 in total) were developed by Dr. Jack Baker for PEER. Additional information about these motions is available at the website: http://peer.berkeley.edu/transportation/projects/ground-motion-studies-for-transportation-systems/

Motion Set 4: These motions (260 in total) include the above **Set2** and **Set3** as well as the additional Bin NEAR of **Set1**.

Once an input motion data set is specified, the user interface will extract/calculate Intensity Measures (IMs) for each of these motions. In total, 11 different Intensity Measures are defined for each motion (and presented to the user in table and graphical forms), including quantities such as Peak Ground Acceleration (PGA), Peak Ground Velocity (PGV), Arias Intensity (AI), and so forth.

7.1.2 Specifications of Input Motions

To conduct a ground shaking analysis, input motions must be defined Fig. 91). The window to define input motions is shown in Fig. 92. To select all motions, click **Select All**. To un-select all motions, click **De-select All.** To remove one motion, select the motion by clicking on it and then click **Delete**. To remove all motions, click **Remove All**.

To add a user-defined motion, click **Import** and then follow the simple steps to import a new motion (Fig. 93). The resulting motion will be added to the current suites of input motion. To obtain a complete new set of input motions, use **Delete All** to remove all existing input motions, and then use **Import** to add new motions.

To import a ground motion file, first save the ground acceleration time history (easy in a notepad .txt file) with each new line being the next acceleration time step. This data in this file should have the acceleration units of g.

The finite element computations can be conducted for several earthquakes at a time. This is employed by specifying **Number of Motions Running Simultaneously** (Fig. 92). You can select as many as 8 records to be run at the same time in order to reduce the overall run time (for dual core machines or better).

Click **View Motion** to view the intensity measures and response spectra of the input motion being highlighted (Fig. 94). SRSS stands for Square Root of the Sum of the Squares of the 2 horizontal components. Click **Display Intensity Measures** to view the intensity measures of the input motion being highlighted (Fig. 95). The user can copy and paste the intensity measures to their favorite text editor such as MS Excel (in Fig. 95, right-click and then click **Select All** (ctrl a) to highlight, and then right-click and then click **Copy** (ctrl c) to copy to the clipboard).

Click **View Histograms & Cumulative Distribution** to view the histogram and cumulative distribution plots for whole input motion set (Fig. 96). The intensity measures include:

- PGA (Peak Ground Acceleration)
- PGV (Peak Ground Velocity)

- PGD (Peak Ground Displacement)
- $D_{5\text{-}95}$ (Strong Motion Duration)
- CAV (Cumulative Absolute Velocity)
- Arias Intensity
- SA (Spectral Acceleration; assuming 1 second period)
- SV (Spectral Velocity; assuming 1 second period)
- SD (Spectral Displacement; assuming 1 second period)
- PSA (Pseudo-spectral Acceleration)
- PSV (Pseudo-spectral Velocity)

The strong motion duration ($D_{5\text{-}95}$) is defined according to the time domain bounded by the 5% and 95% cumulative Arias intensity of the record. All of the spectral intensity measures are defined at an effective viscous damping of 5% unless otherwise noted.

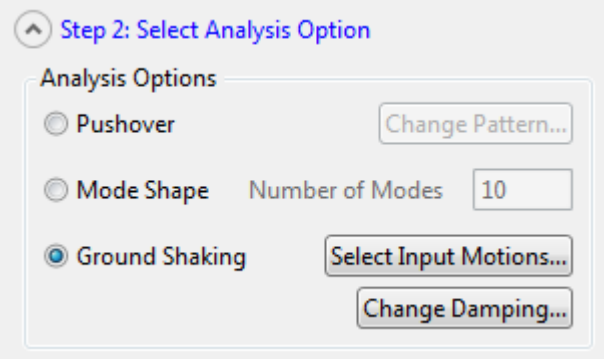


Fig. 91. Group shaking analysis

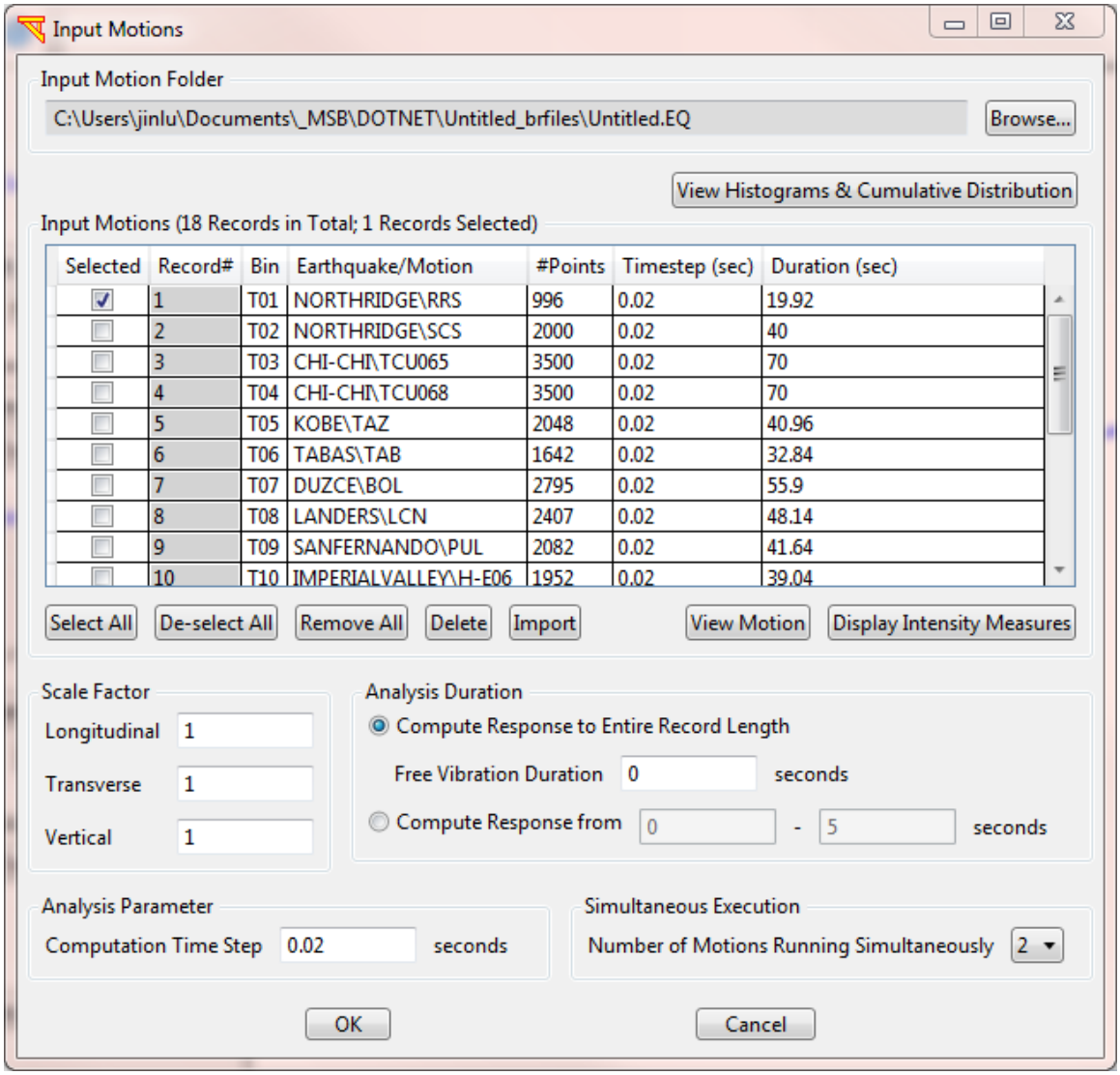
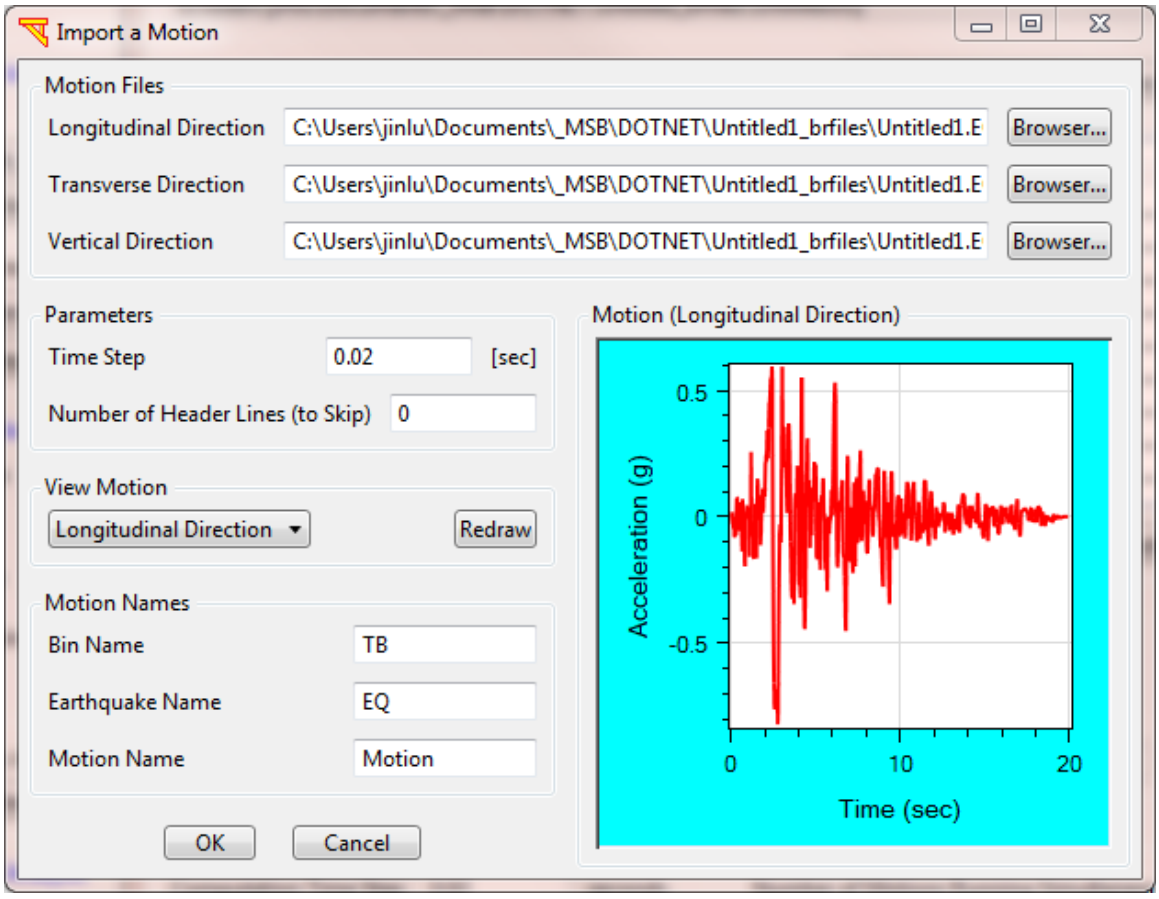
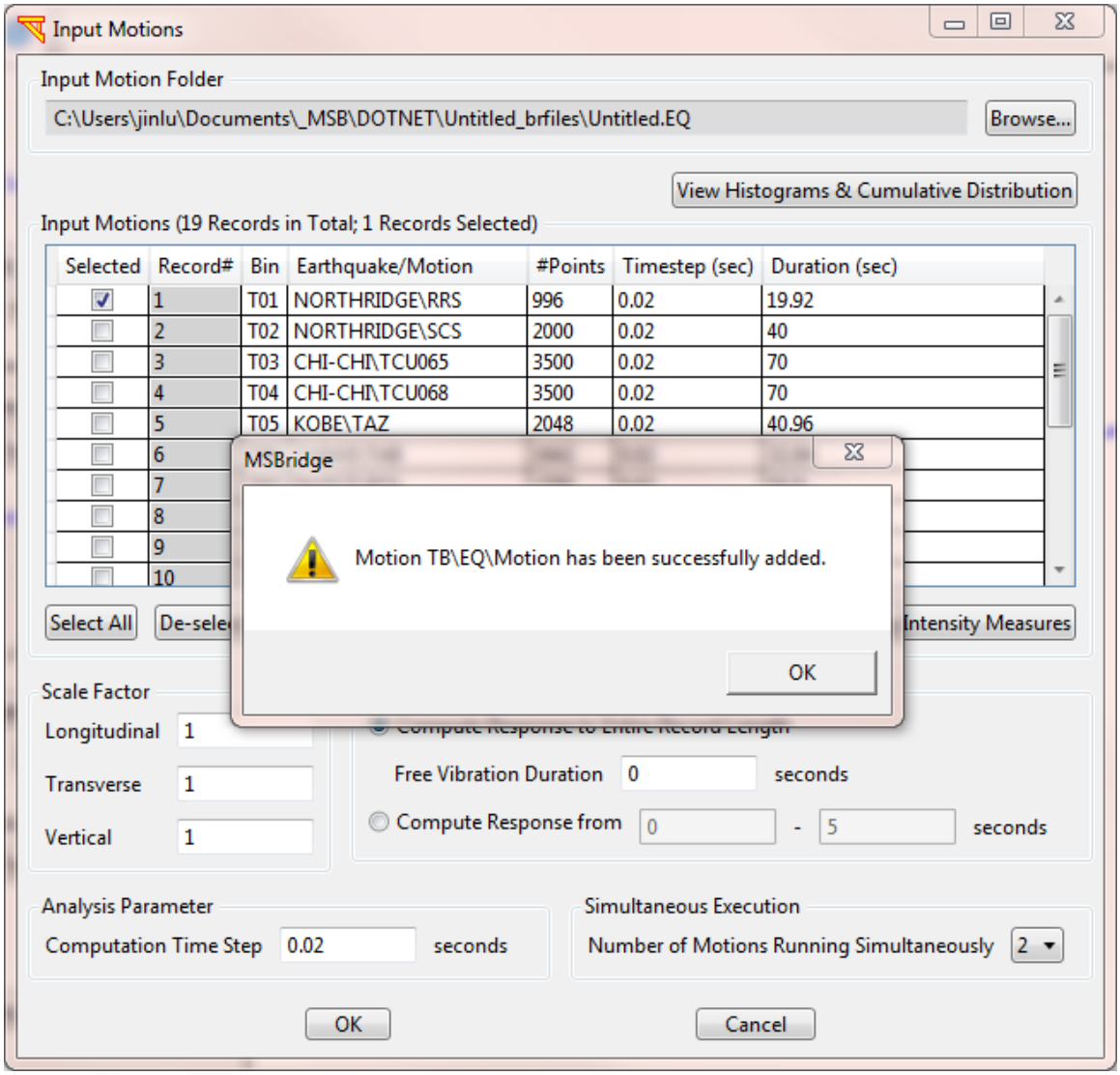


Fig. 92. Input motions window


Import a Motion
Motion Files
Longitudinal Direction
C:\Users\jinlu\Documents_MSB\DOTNET\Untitled1_brfiles\Untitled1.E
Browser...
Transverse Direction
C:\Users\jinlu\Documents_MSB\DOTNET\Untitled1_brfiles\Untitled1.E
Browser...
Vertical Direction
C:\Users\jinlu\Documents_MSB\DOTNET\Untitled1_brfiles\Untitled1.E
Browser...
Parameters
Time Step
0.02
[sec]
Number of Header Lines (to Skip)
0
View Motion
Longitudinal Direction
Redraw
Motion Names
Bin Name
TB
Earthquake Name
EQ
Motion Name
Motion
OK
Cancel
Motion (Longitudinal Direction)
0.5
0
-0.5
Acceleration (g)
0
10
20
Time (sec)

a)



b)

Fig. 93. Importing a user-defined motion: a) choosing data files; b) message showing new motion has been added

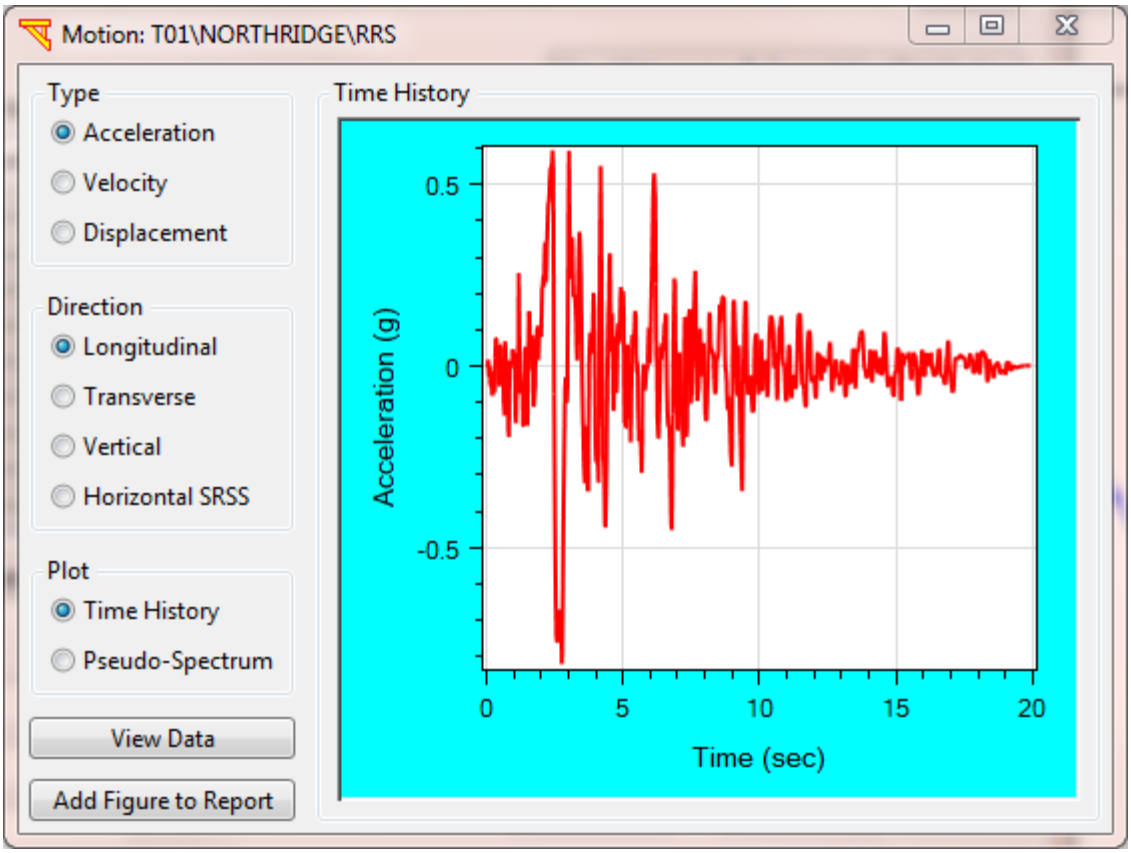
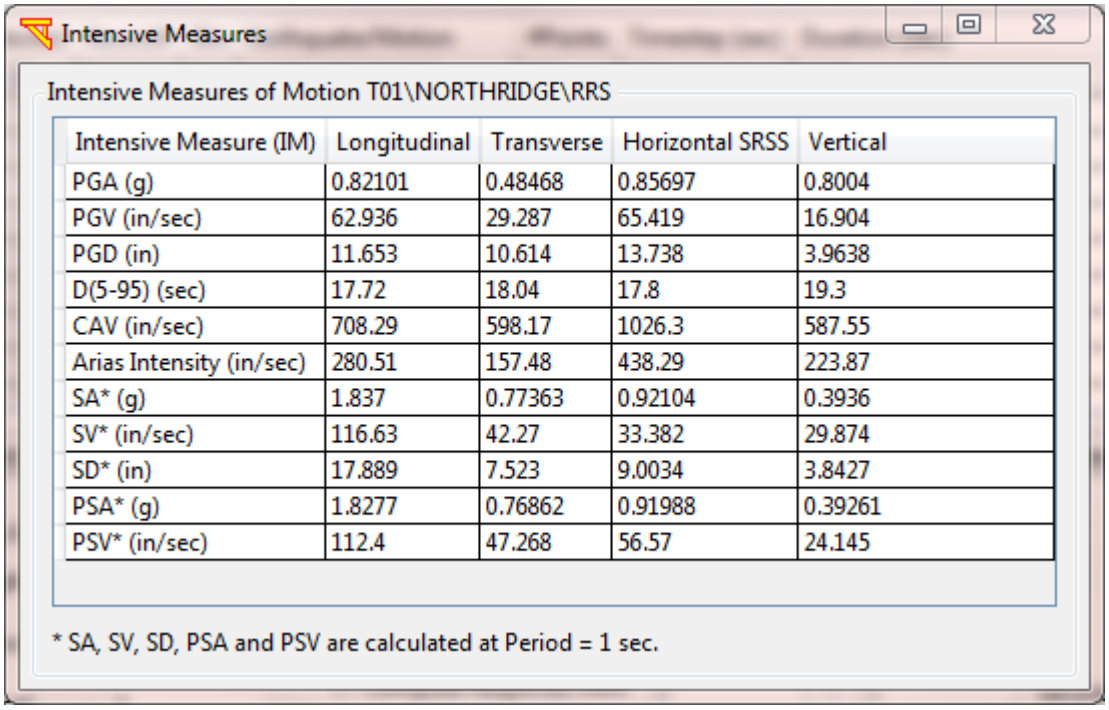
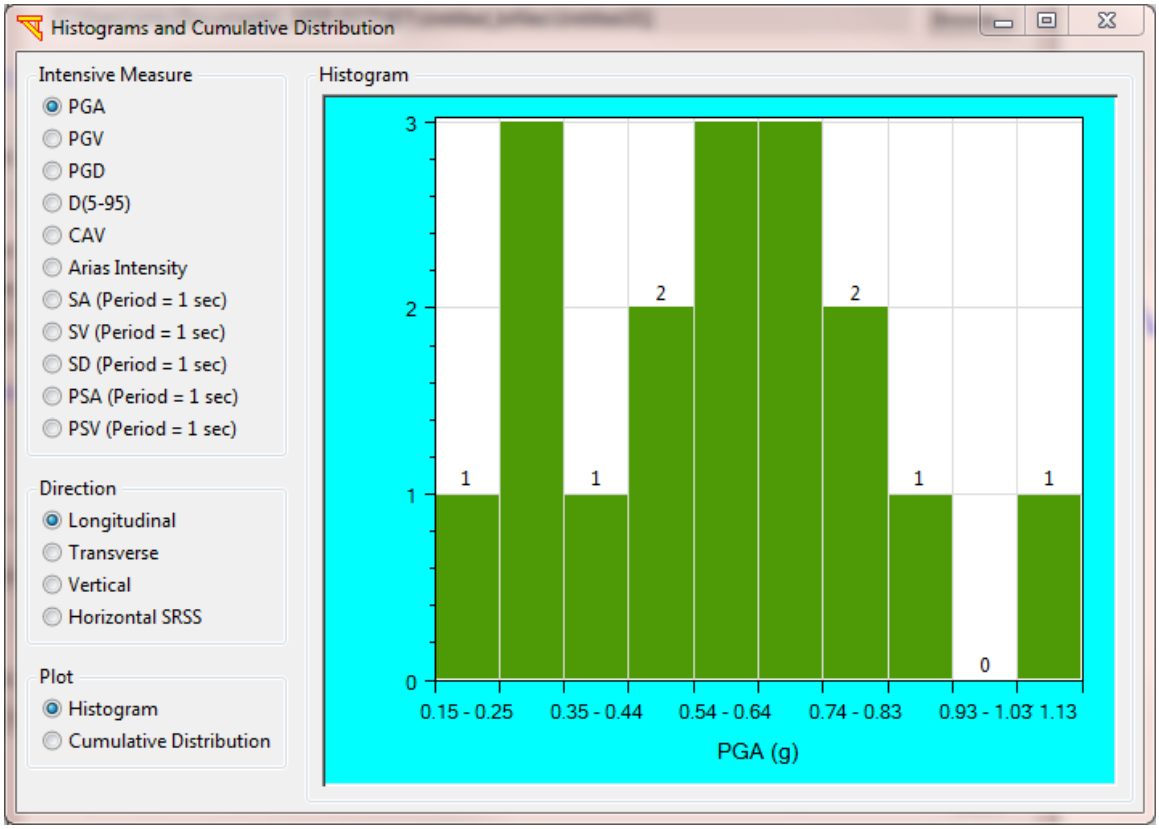


Fig. 94. Time histories and response spectra of individual motion

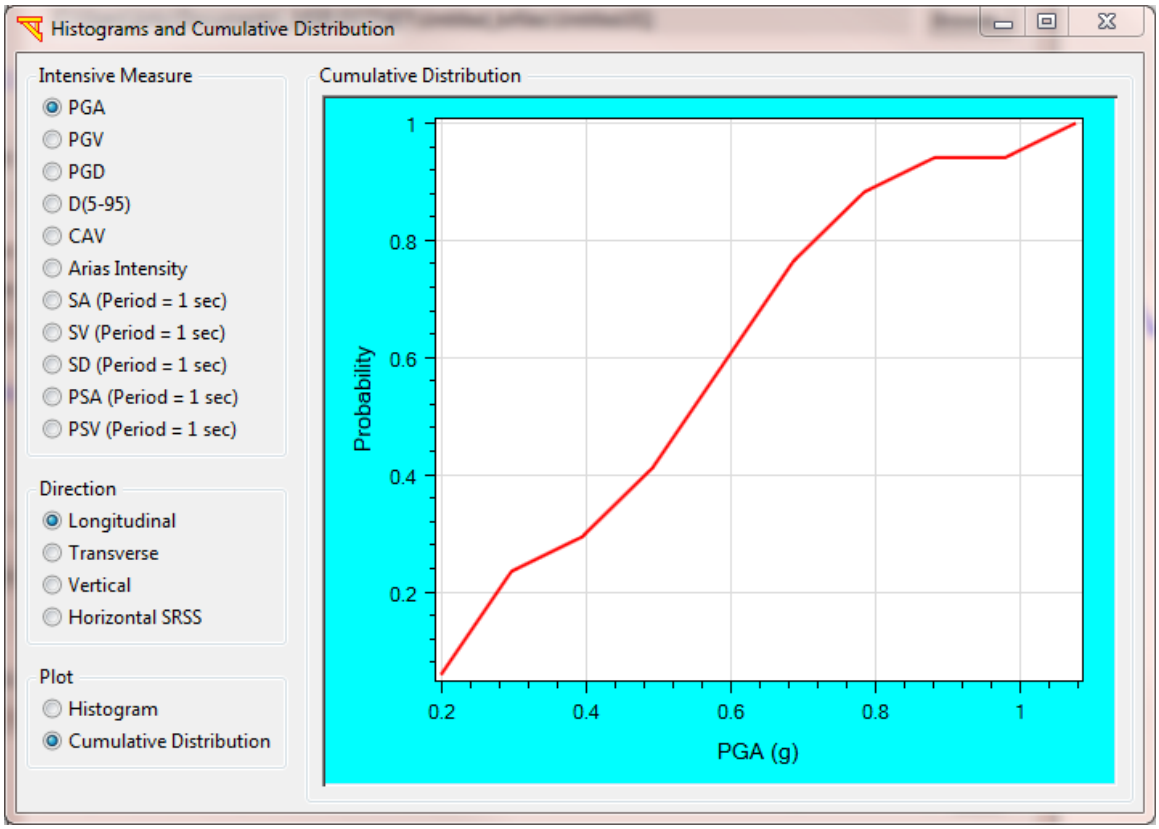


Intensive Measure (IM)	Longitudinal	Transverse	Horizontal SRSS	Vertical
PGA (g)	0.82101	0.48468	0.85697	0.8004
PGV (in/sec)	62.936	29.287	65.419	16.904
PGD (in)	11.653	10.614	13.738	3.9638
D(5-95) (sec)	17.72	18.04	17.8	19.3
CAV (in/sec)	708.29	598.17	1026.3	587.55
Arias Intensity (in/sec)	280.51	157.48	438.29	223.87
SA* (g)	1.837	0.77363	0.92104	0.3936
SV* (in/sec)	116.63	42.27	33.382	29.874
SD* (in)	17.889	7.523	9.0034	3.8427
PSA* (g)	1.8277	0.76862	0.91988	0.39261
PSV* (in/sec)	112.4	47.268	56.57	24.145

Fig. 95. Intensity measures of individual motion


Histograms and Cumulative Distribution
Intensive Measure
PGA
PGV
PGD
D(5-95)
CAV
Arias Intensity
SA (Period = 1 sec)
SV (Period = 1 sec)
SD (Period = 1 sec)
PSA (Period = 1 sec)
PSV (Period = 1 sec)
Direction
Longitudinal
Transverse
Vertical
Horizontal SRSS
Plot
Histogram
Cumulative Distribution
Histogram
3
2
1
0
1
1
2
2
1
0
1
0.15 - 0.25
0.35 - 0.44
0.54 - 0.64
0.74 - 0.83
0.93 - 1.03
1.13
PGA (g)

a)



b)

Fig. 96. Histogram and cumulative distribution for the whole input motion set: a) histogram; b) cumulative distribution

7.1.3 Rayleigh Damping

MSBridge employs Rayleigh damping, which takes the form:

$\mathbf{C} = A_m \mathbf{M} + A_k \mathbf{K}$

where **M** is the mass matrix, **C** is the viscous damping matrix, **K** is the initial stiffness matrix. A_m and A_k are two user-specified constants.

The damping ratio curve $\xi(f)$ is calculated based on the following equation:

$$\xi = \frac{A_m}{4\pi f} + A_k \pi f$$

where f is frequency.

(1) Specification of A_m and A_k By Defining Damping Ratios

Click **Change Damping** in the MSBridge main window to modify the Rayleigh damping coefficients (Fig. 97). The user can define damping coefficients (Fig. 97) by specifying two frequencies, f_1 and f_2 (must be between 0.1 and 50 Hz), and two damping ratios, ξ_1 and ξ_2 (suggested values are between 0.2% and 20%).

The Rayleigh damping parameters A_m and A_k are obtained by solving the follow equations simultaneously:

$$\xi_1 = \frac{A_m}{4\pi f_1} + A_k \pi f_1$$

$$\xi_2 = \frac{A_m}{4\pi f_2} + A_k \pi f_2$$

(2) Direct Specification of A_m and A_k:

The user can also directly define Rayleigh damping coefficients A_m and A_k (Fig. 97).

7.2 Save Model and Run Analysis

After defining the finite element model, click **Save Model and Run Analysis**. The finite element computations will start, for several earthquakes at a time (Fig. 98) as specified in the **Input Motions** window (Fig. 92).

The user can modify the time integration scheme for the OpenSees analysis by clicking Menu **Execute** and then **Advanced Option: OpenSees Parameters** (Fig. 99). Fig. 99 shows the default parameters which are used in the analysis.

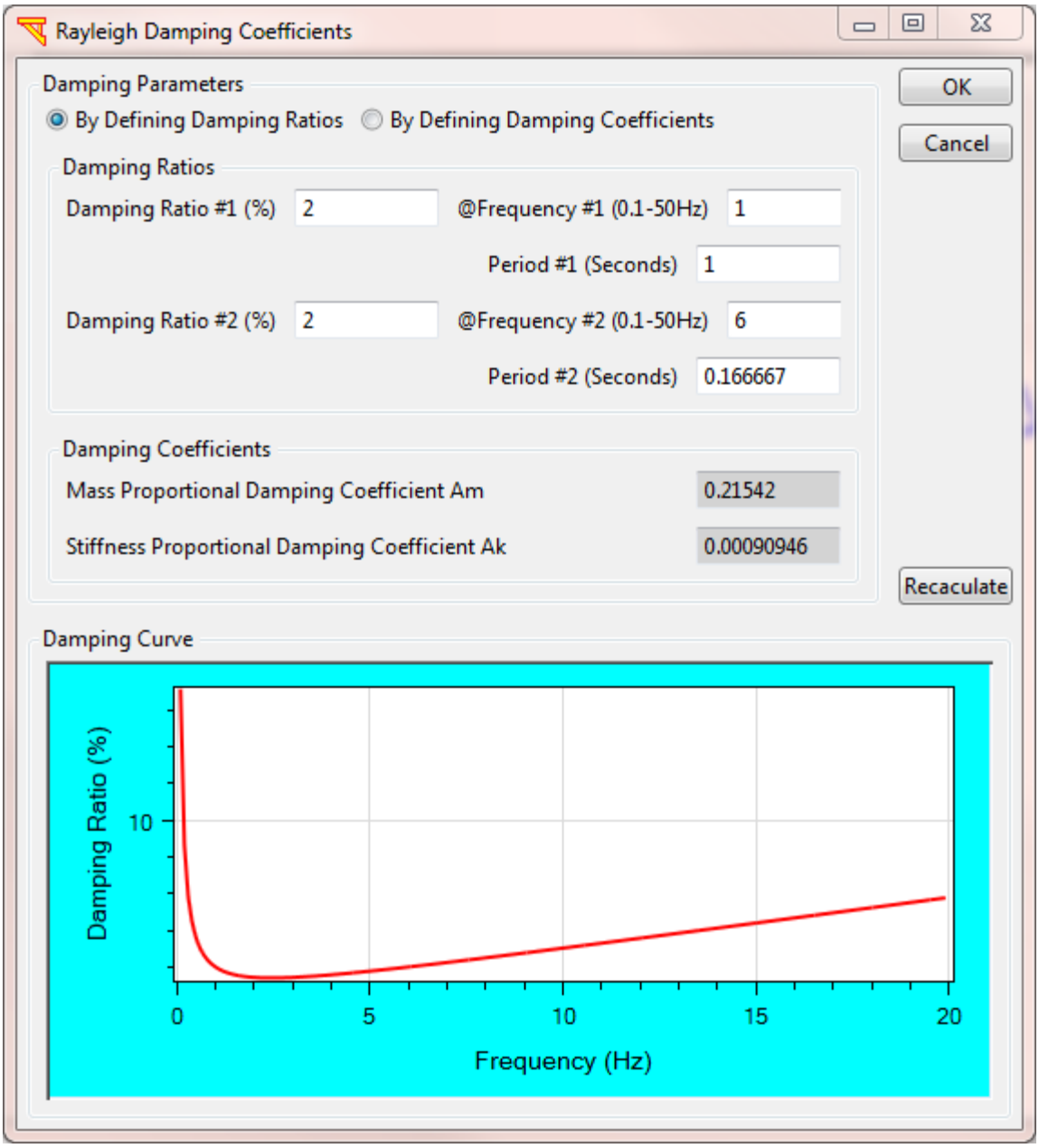


Fig. 97. Rayleigh damping

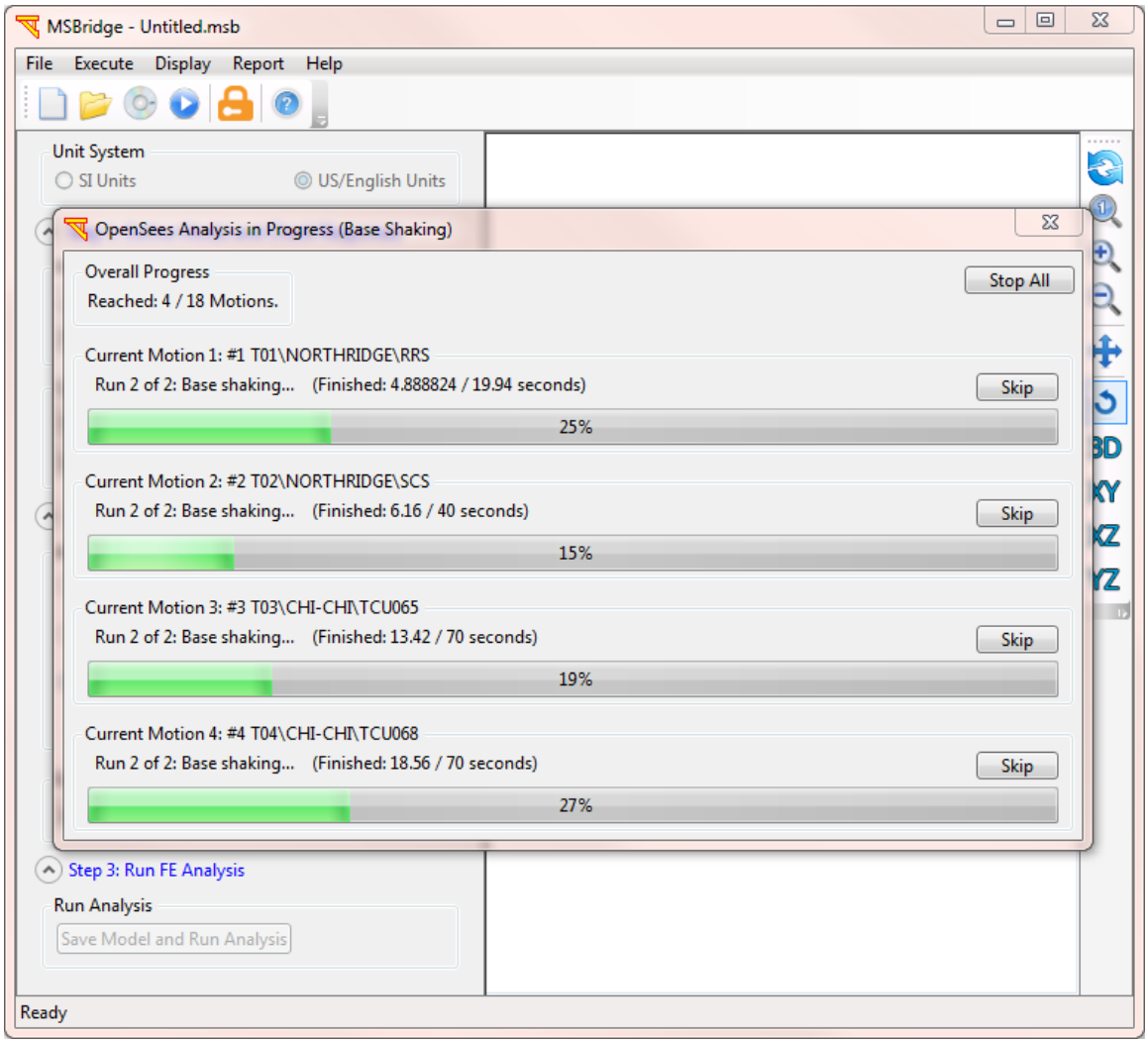


Fig. 98. Simultaneous execution of analyses for multiple motions

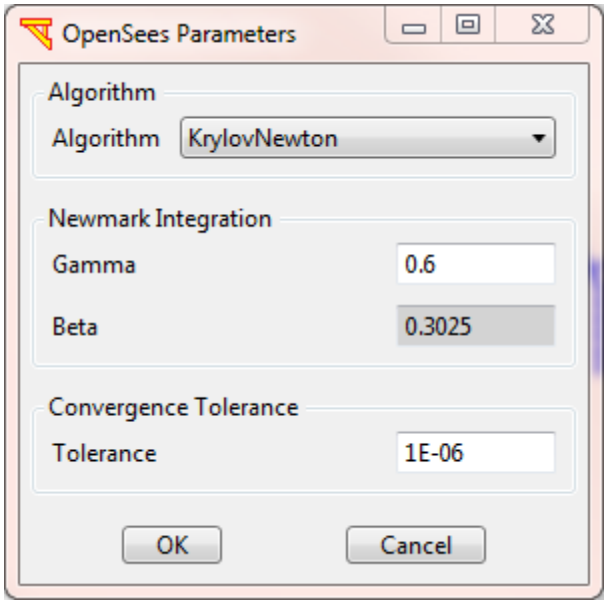


Fig. 99. Parameters for OpenSees analysis

8 Time History and Engineering Demanding Parameter Output

8.1 Time History Output Quantities

At the end of the FE analysis phase, time histories and bridge responses will be available of the form:

i) Column Response Time Profiles
ii) Column Response Time Histories
iii) Column Response Relationships
iv) Abutment Responses
v) Deformed Mesh

In addition, for multiple earthquake analysis scenarios, Intensity Measures (IMs) and response spectra for each input motion are calculated and are available for display in Table and Figure formats. Engineering Demand Parameter (EDP) Quantities and Bridge peak accelerations for all employed shaking motions are also available for display against any of the computed IMs.

The post-processing capabilities can be accessed from Menu **Display** (Fig. 3). To display output for a different input motion, click Menu **Display** and then **Detailed Output: Please Select Input Motion** (Fig. 3d). The name of the selected input motion will also appear on the menu items (Fig. 3d).

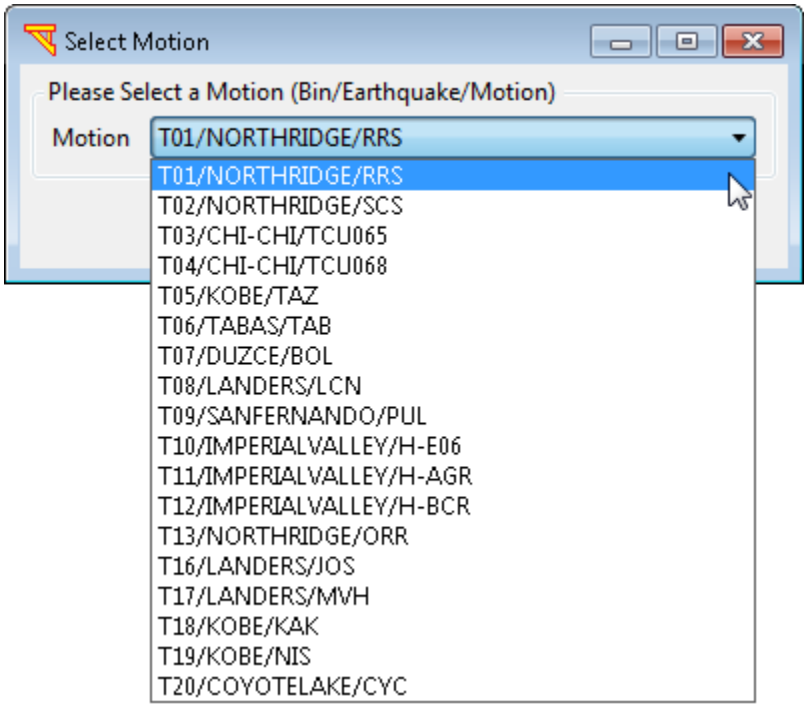


Fig. 100. Selection of an input motion

8.1.1 Deck Response Time Histories

The deck response time histories can be accessed by clicking menu **Display** (Fig. 3) and then **Deck Response Time Histories**. Fig. 101 shows the window for displaying the deck longitudinal displacement time histories.

8.1.2 Column Response Profiles

The column response profiles can be accessed by clicking menu **Display** (Fig. 3) and then **Column Response Profiles**. The column response window is shown in Fig. 102. The columns are labeled as:

i) **Column 1 of Bent 2** (see Fig. 1., the first bent starting after left abutment is denoted as "Bent 2", the second as "Bent 3", and so on)
ii) Column 2 of Bent 2
iii) (more if any)

Fig. 103 shows the bending moment in the longitudinal plane. The horizontal axis of the plot is the response name (e.g., displacement, bending moment, etc.) and the vertical axis is the elevation of the column. Zero elevation means the column base.

8.1.3 Column Response Time Histories

The column response time histories can be accessed by clicking menu **Display** (Fig. 3) and then **Column Response Time Histories**. Fig. 104 shows the window for displaying the column longitudinal displacement time histories.

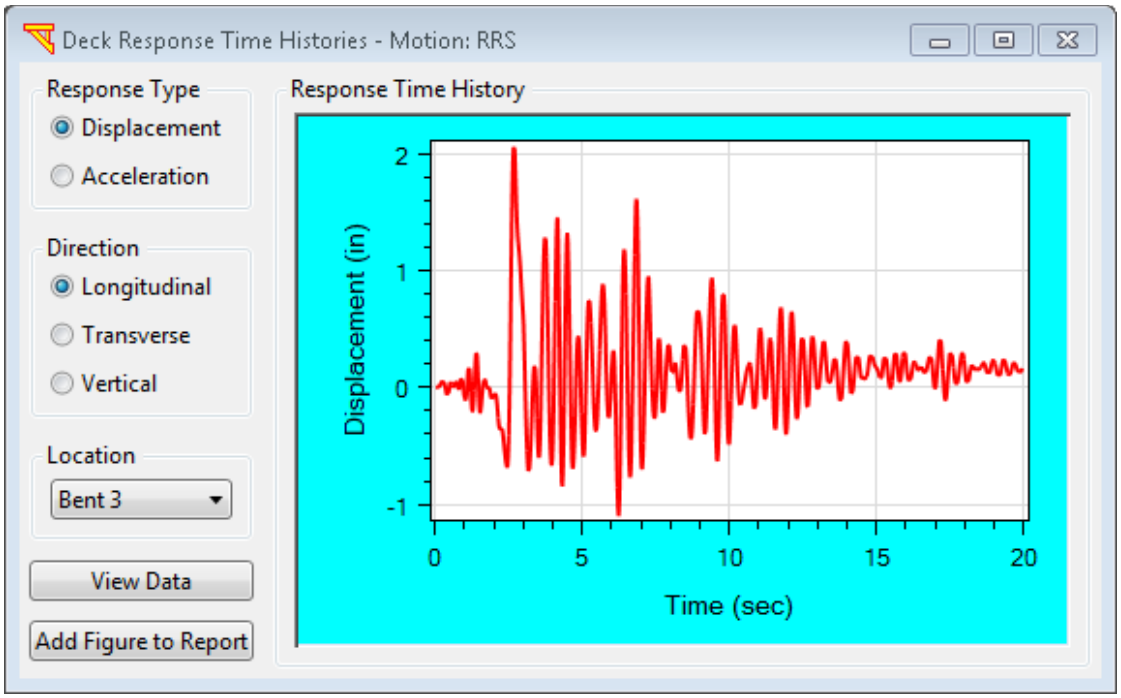


Fig. 101. Deck longitudinal displacement response time histories

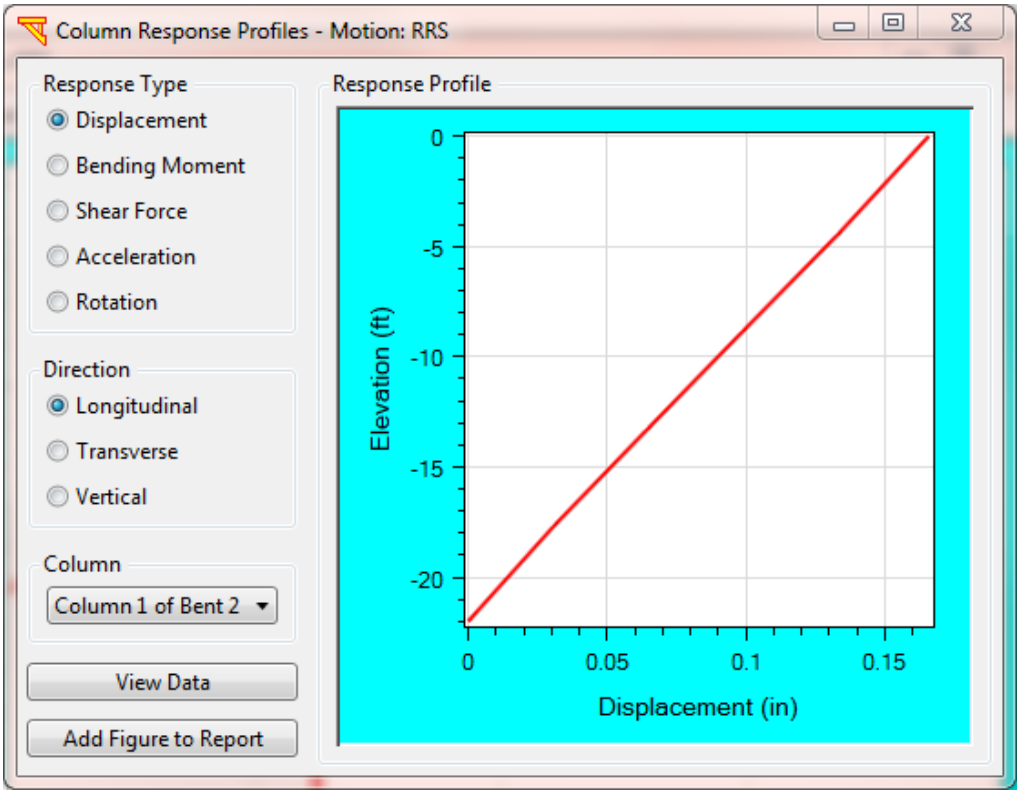


Fig. 102. Displacement profile in the longitudinal plane

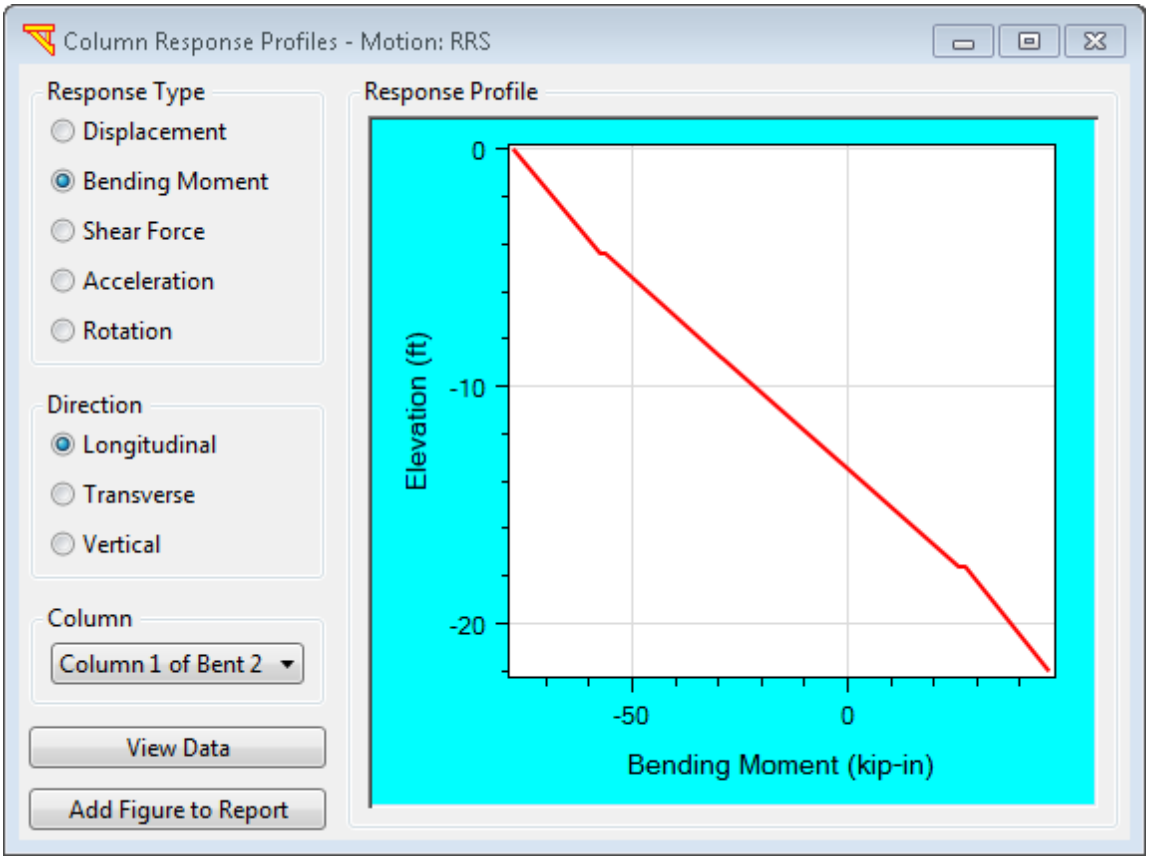


Fig. 103. Bending moment profile in the longitudinal plane

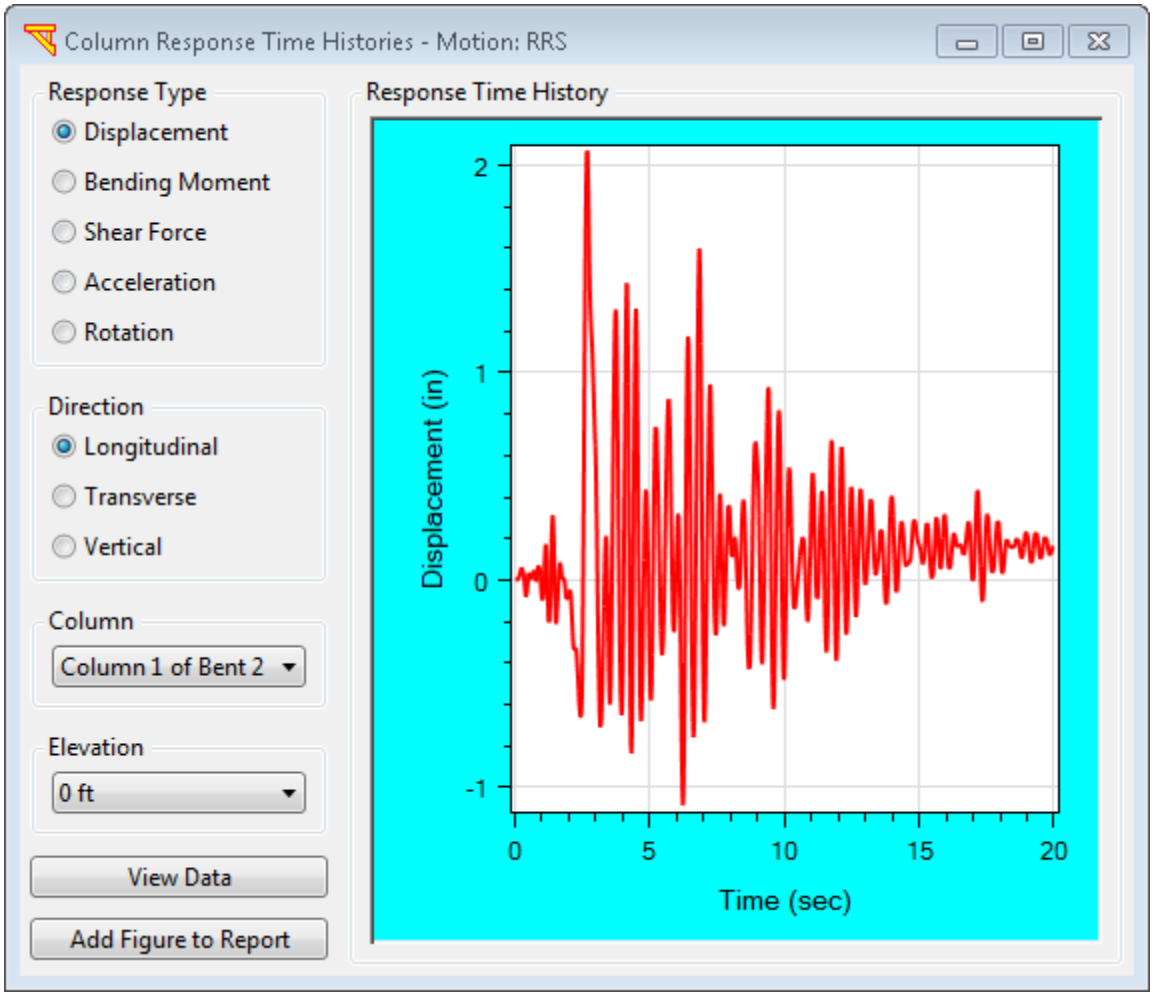


Fig. 104. Response time histories and profiles for column (and pile shaft): displacement is shown at the nodes.

8.1.4 Column Response Relationships

The column response relationships can be accessed by clicking menu **Display** (Fig. 3) and then **Column Response Relationships**.

The **Elevation** box includes all elevations (starting from column top). Zero elevation refers to the column top.

Fig. 105 shows the longitudinal load-displacement curve at the column top. The load refers to the shear force of the beam-column element at the specified elevation. Fig. 106 shows the moment-curvature curve at the column top. The vertical axis is the bending moment and the horizontal axis is the curvature. To view the data for the plot, click **View Data**.

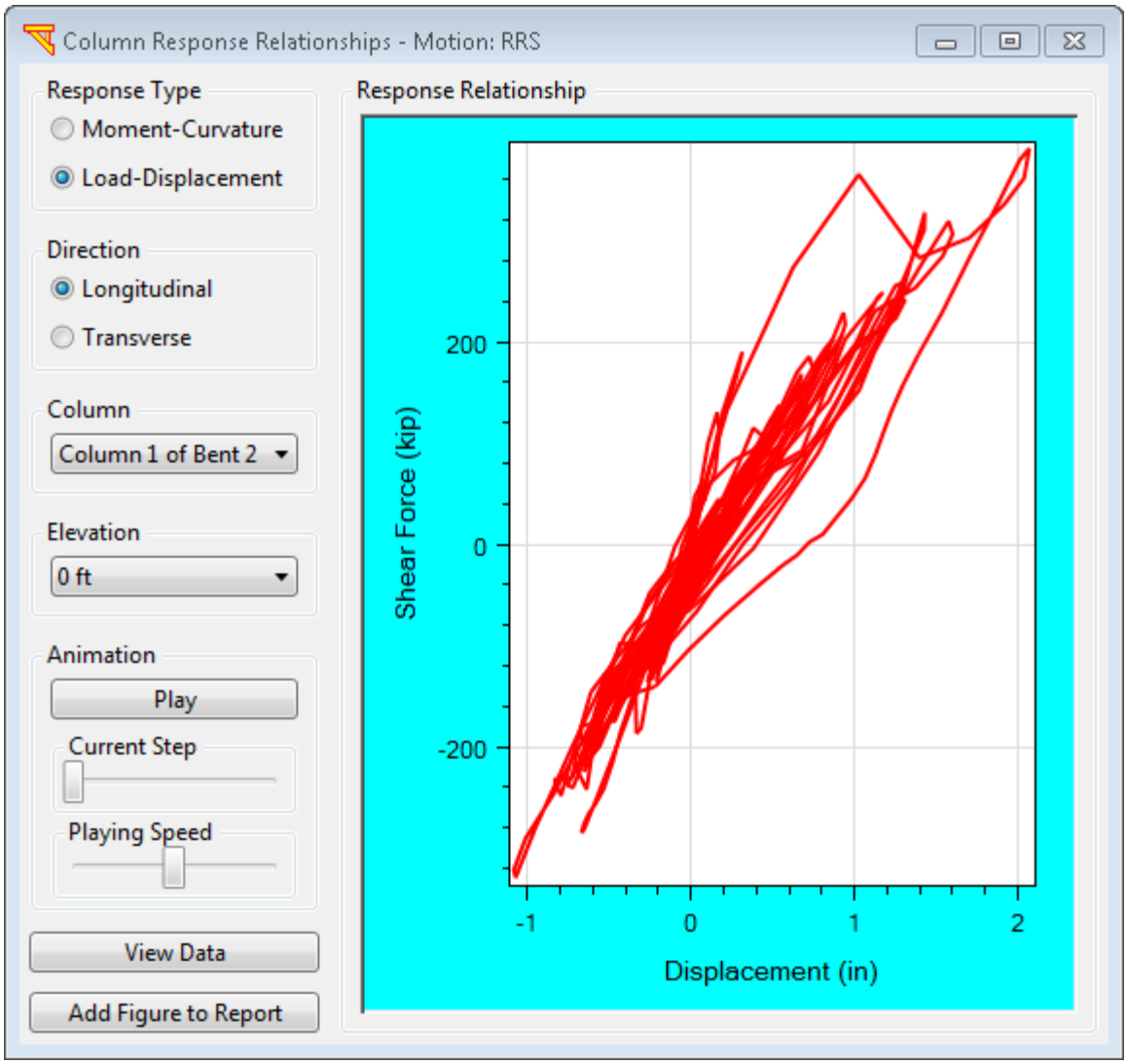


Fig. 105. Load-displacement curve at column top

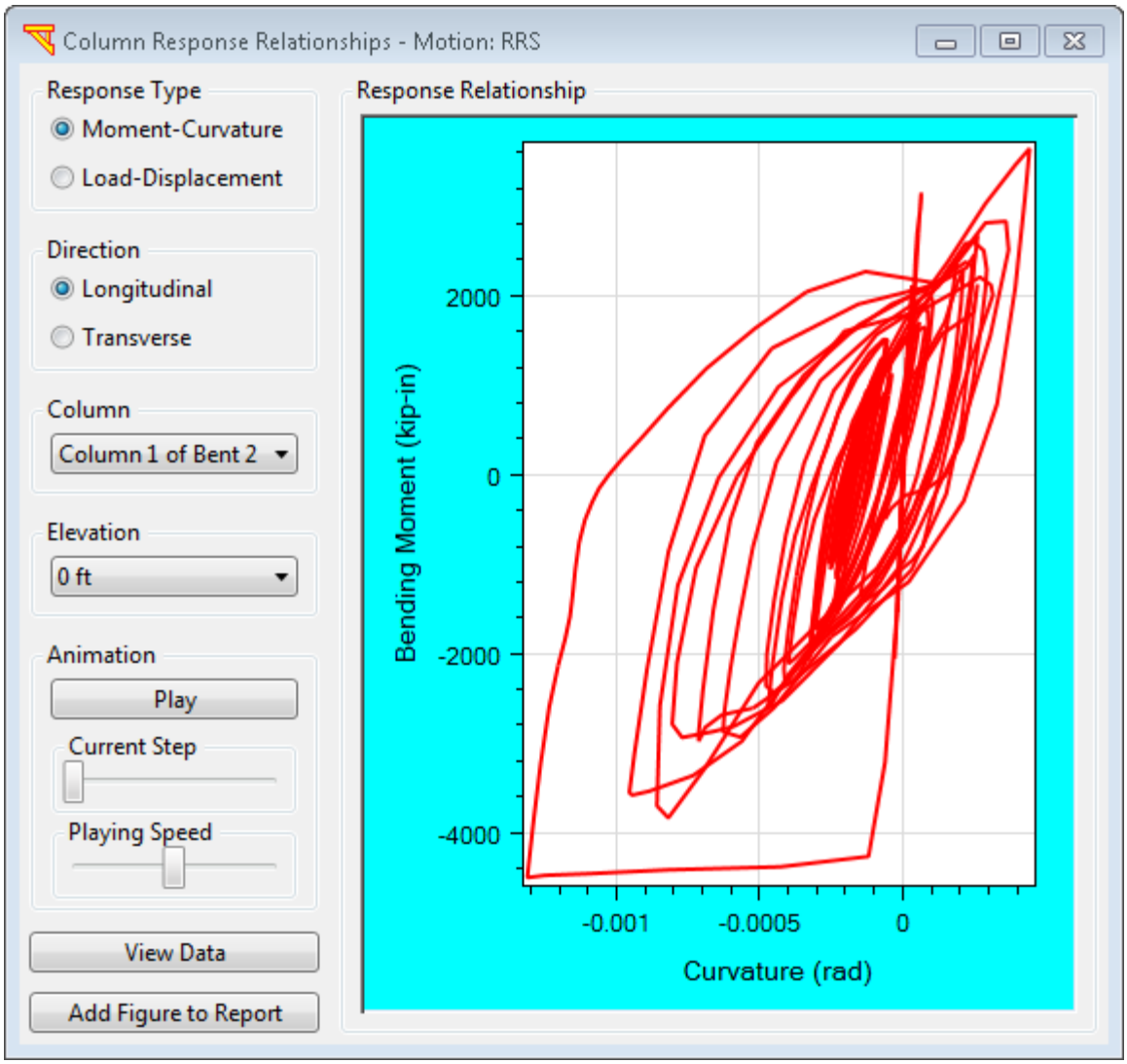


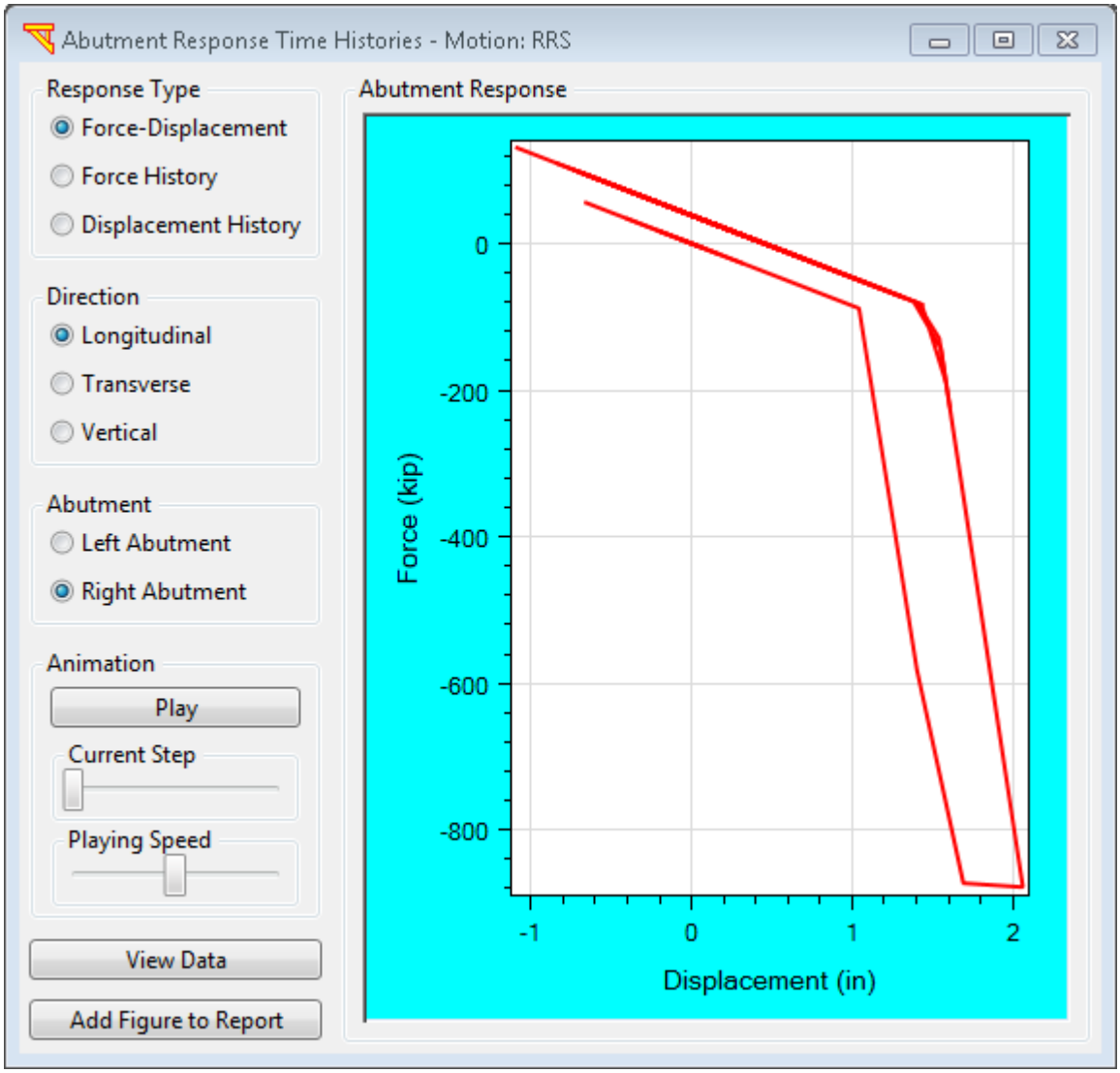
Fig. 106. Moment-curvature curve at column top

8.1.5 Abutment Responses Time Histories

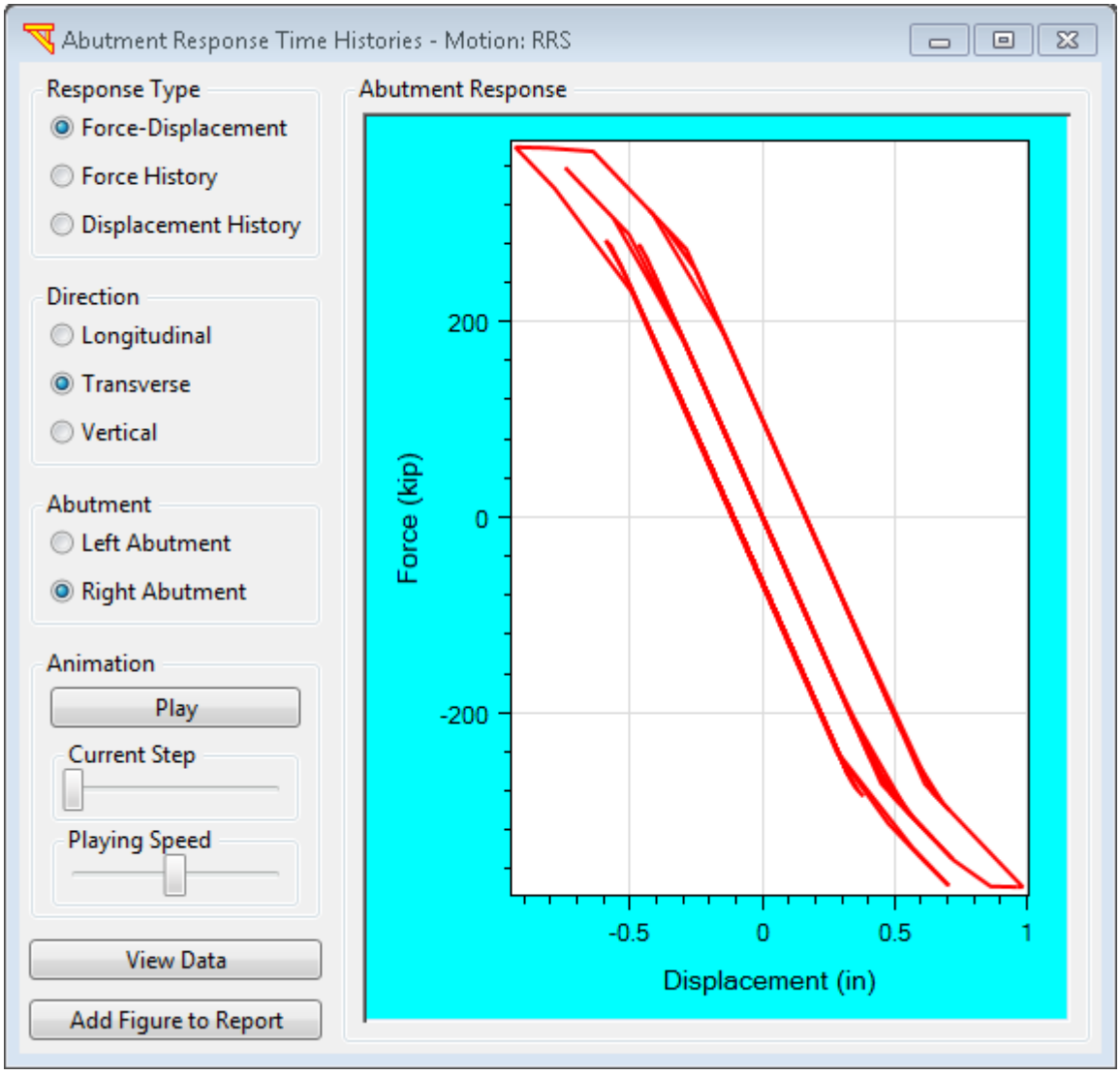
The abutment responses can be accessed by clicking menu **Display** and then **Abutment Response Time Histories**. The abutment responses window includes the following options:

i) **Force-Displacement Relationships**
ii) **Relative Deck-end/Abutment Displacement Time Histories**
iii) **Resisting Force Time Histories**

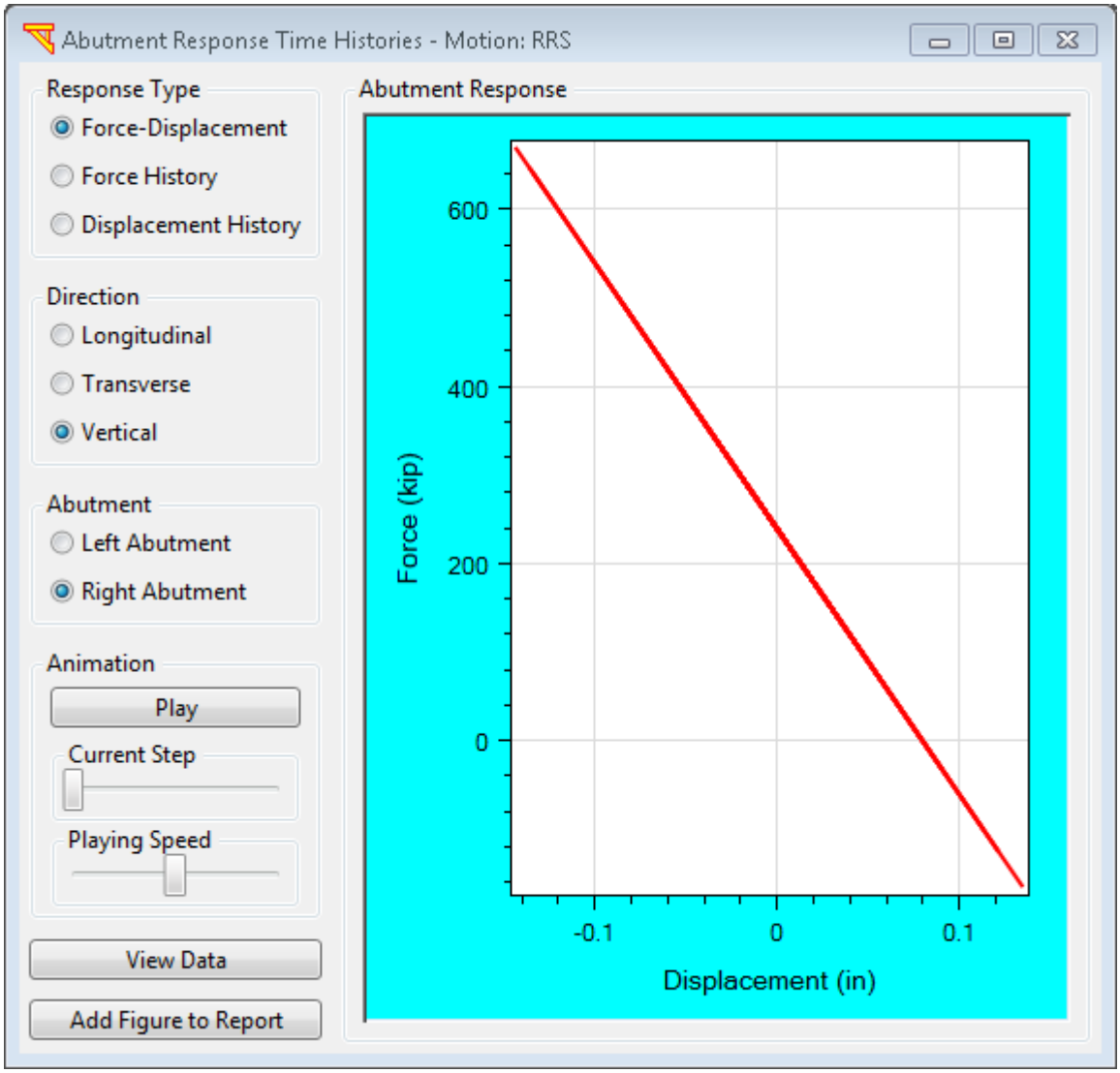
Three directions (longitudinal, transverse and vertical directions) of the above responses for both left and right abutments are all displayed. Fig. 107 shows the abutment response time histories. The force refers to the resisting force acting on deck-end and the displacement refers to the relative deck-end/abutment displacement.


Abutment Response Time Histories - Motion: RRS
Response Type
Force-Displacement
Force History
Displacement History
Direction
Longitudinal
Transverse
Vertical
Abutment
Left Abutment
Right Abutment
Animation
Play
Current Step
Playing Speed
View Data
Add Figure to Report
Abutment Response
0
-200
-400
-600
-800
Force (kip)
-1
0
1
2
Displacement (in)

a)


Abutment Response Time Histories - Motion: RRS
Response Type
Force-Displacement
Force History
Displacement History
Direction
Longitudinal
Transverse
Vertical
Abutment
Left Abutment
Right Abutment
Animation
Play
Current Step
Playing Speed
View Data
Add Figure to Report
Abutment Response
200
0
-200
Force (kip)
-0.5
0
0.5
1
Displacement (in)

b)



c)

Fig. 107. Abutment response force-displacement relationships: a) longitudinal response; b) transverse response; and c) vertical response

8.1.6 Soil Spring Responses Time Histories

The soil spring responses can be accessed by clicking menu **Display** and then **Soil Spring Response Time Histories**. The soil spring responses window includes the following options (Fig. 108):

i) **Force-Displacement Curve**
ii) **Displacement Time History**
iii) **Force Time History**

Two directions (longitudinal and transverse directions) of the above responses for each soil spring are all displayed (Fig. 108).

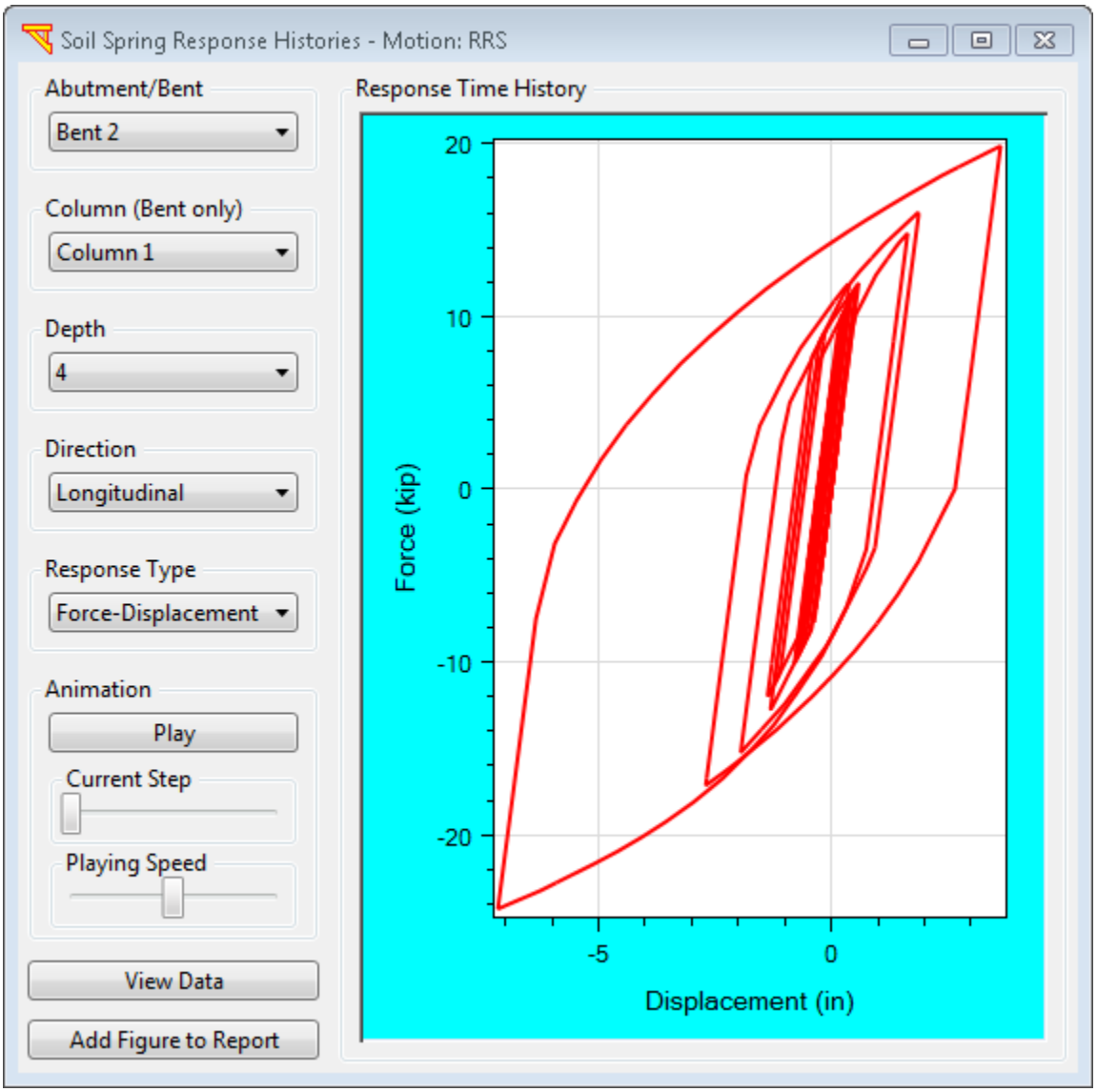


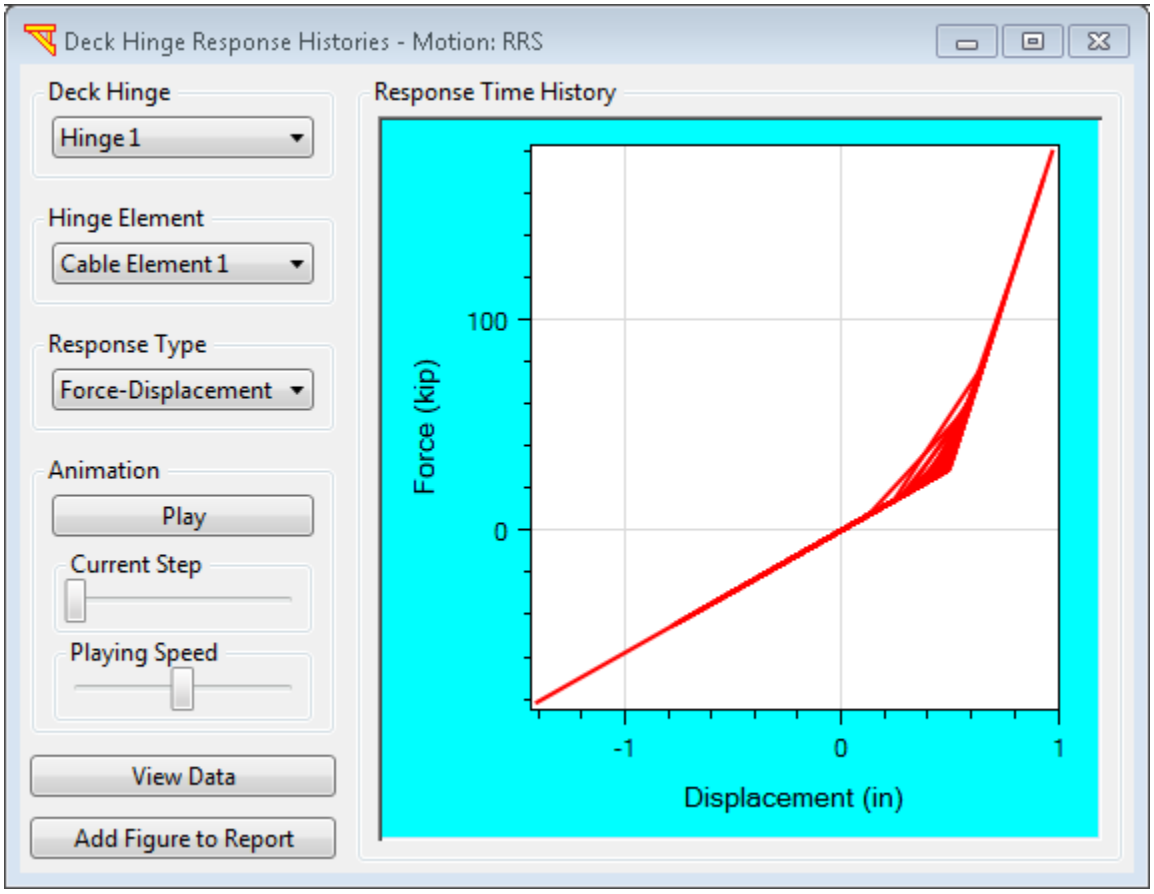
Fig. 108. Soil spring response time histories

8.1.7 Deck Hinge Responses Time Histories

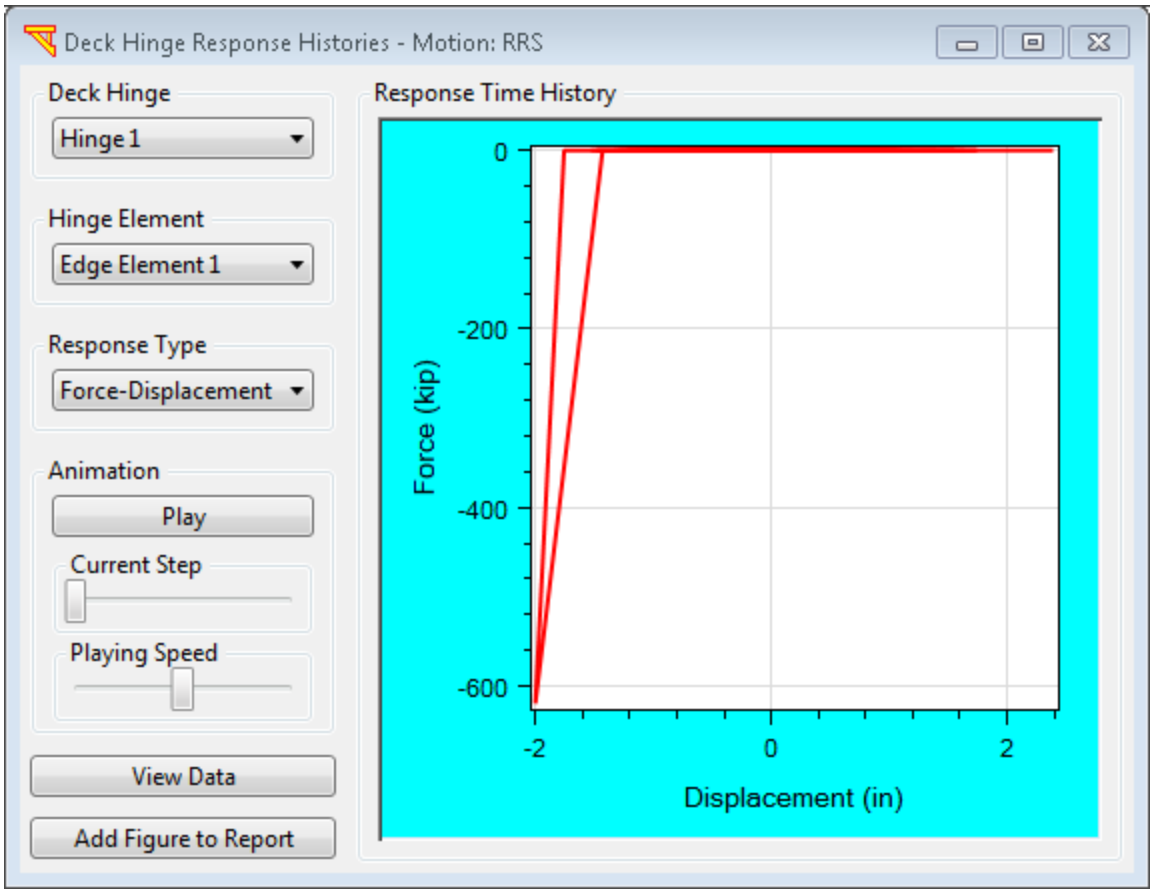
The deck hinge responses can be accessed by clicking menu **Display** and then **Deck Hinge Response Time Histories**. The deck hinge responses window includes the following options (Fig. 109):

i) **Force-Displacement Curve**
ii) **Displacement Time History**
iii) **Force Time History**

Response time histories are shown for the cable and edge hinge elements for each hinge (Fig. 109).



a)



b)

Fig. 109. Deck hinge response time histories: a) cable element; b) edge element

8.1.8 Isolation Bearing Responses Time Histories

The isolation bearing responses can be accessed by clicking menu **Display** and then **Isolation Bearing Response Time Histories**. The isolation bearing responses window includes the following options (Fig. 110):

i) **Force-Displacement Curve**
ii) **Displacement Time History**
iii) **Force Time History**

Three translational directions and three rotational directions of the above responses for each bearing are displayed (Fig. 110).

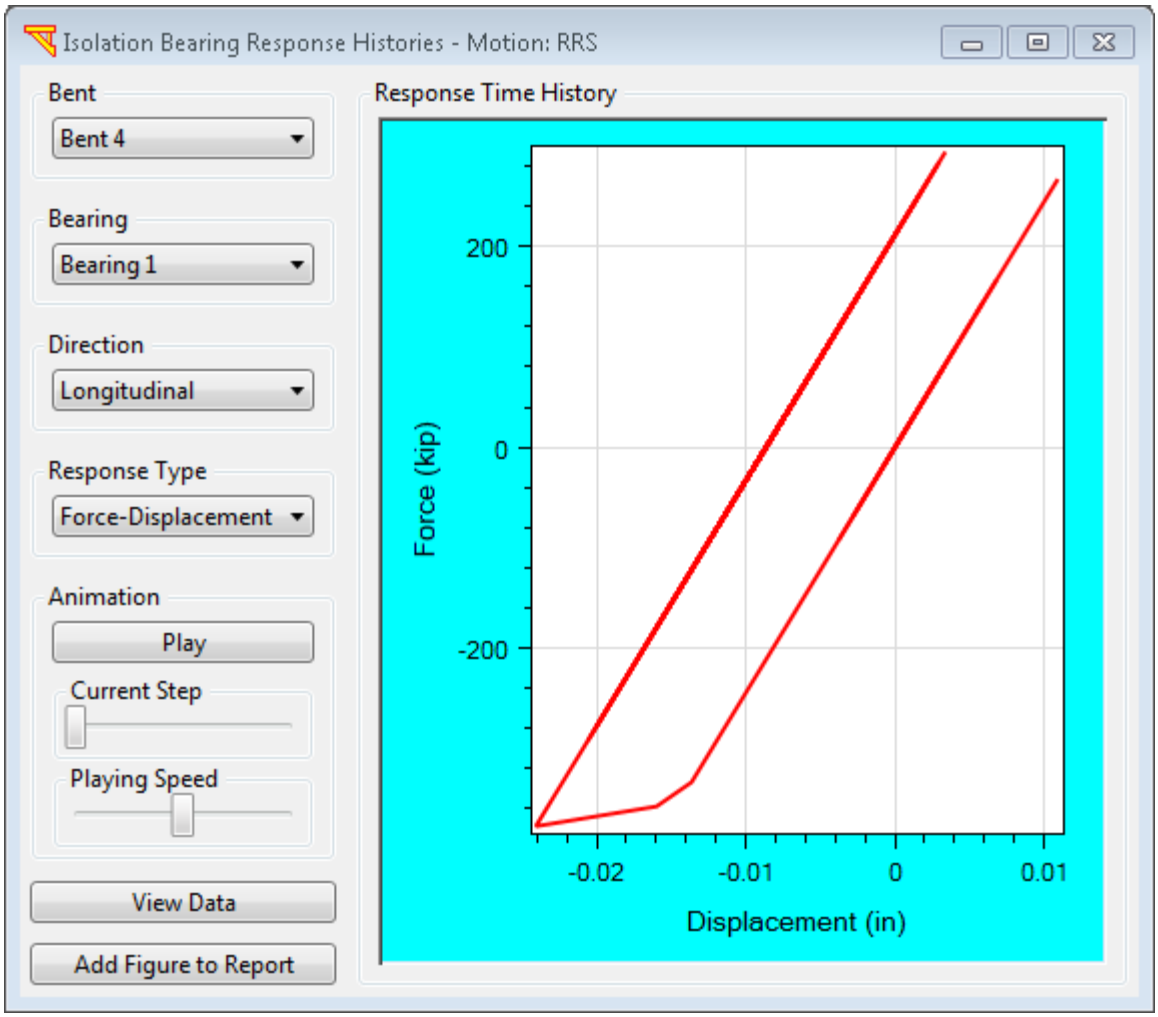


Fig. 110. Isolation bearing response time histories

8.2 Deformed Mesh and Animation

The deformed mesh can be accessed by clicking menu **Display** (Fig. 3) and then **Deformed Mesh**. The deformed mesh window is shown in Fig. 111.

Analysis stages include **Due to gravity** and **Due to pushover** (or **Due to base shaking**). The response types include

- **i) Deformed mesh**
- **ii) Resultant Disp.**
- **iii) X-Displacement**
- **iv) Y-Displacement**
- **v) Z-Displacement**
- **vi) Plastic Hinges**

In the Ground Shaking Analysis, the input motion is also animated at the deformed mesh window along with bridge displacement (Fig. 111).

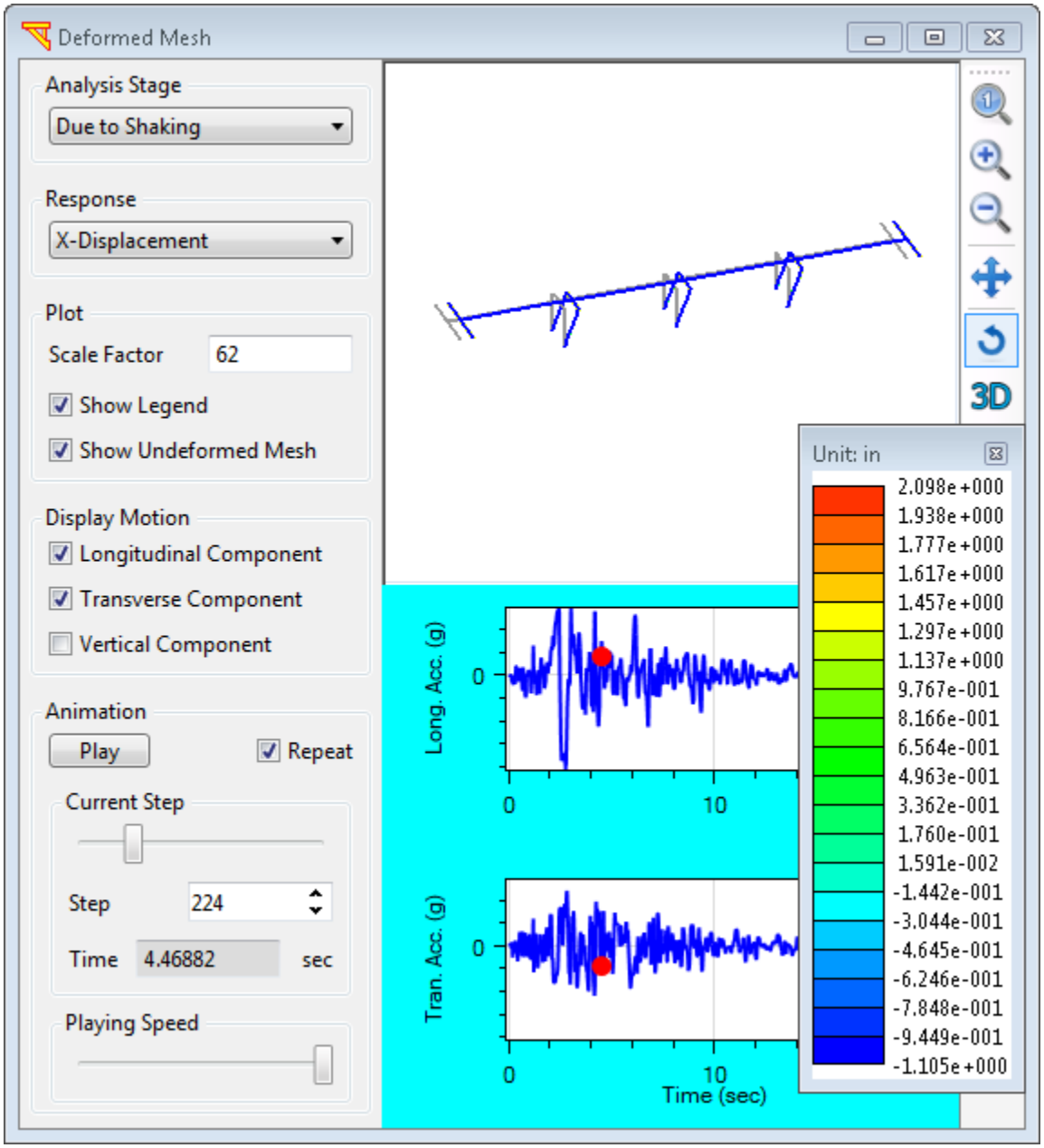


Fig. 111. Deformed mesh

Visualization of plastic hinges is available if the nonlinear beam-column element is used for the columns. In the Ground Shaking Analysis, the input motion is also animated at the deformed mesh window along with the development of plastic hinges (Fig. 112). In the current version, the visualization is implemented in such a way that a plastic hinge

marker stays once the plastic hinge is developed. The plastic hinge is developed when rebar fails in tension or first concrete fiber reaches the maximum strain capacity.

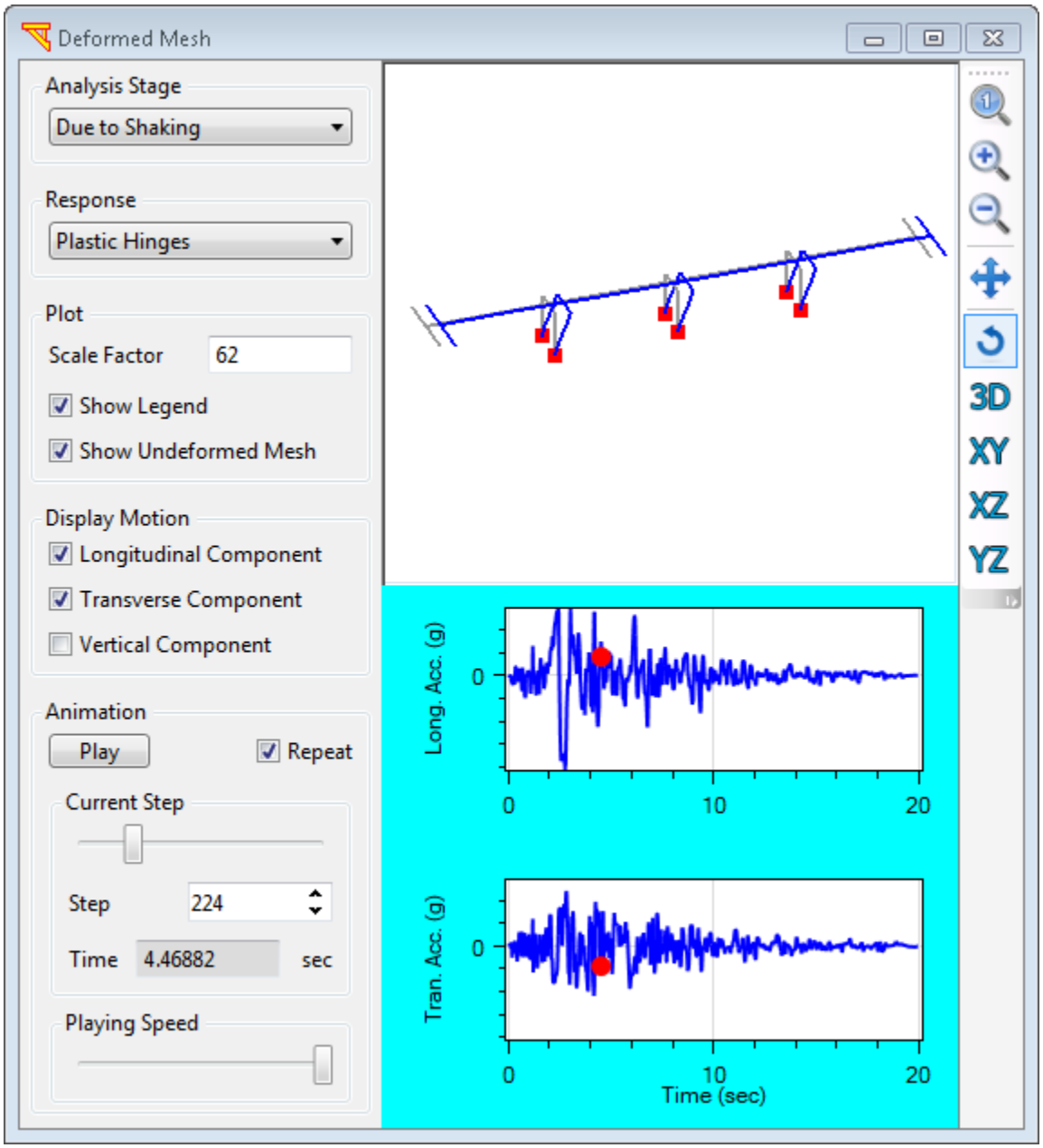


Fig. 112. Visualization of Plastic Hinges

8.3 Maximum Output Quantities

8.3.1 EDP Quantities

At the end of the finite element analysis phase, the following output EDP quantities (for each earthquake record) are available:

Table 6. Engineering Demand Parameters (EDP)

EDP #	EDP names
1	Maximum column drift ratio
2	Residual column drift ratio
3	Maximum relative deck-end/abutment displacement (left)
4	Maximum relative deck-end/abutment displacement (right)
5	Maximum bridge-abutment bearing displacement (left)
6	Maximum bridge-abutment bearing displacement (right)
7	Approach residual vertical displacement (left)
8	Approach residual vertical displacement (right)

The EDP outcomes can be shown against the input base shaking IMs. The sections below detail how the response quantities are obtained for each EDP for the annotated model that is used to describe the location of sampling points during time history analysis.

EDP1: Maximum drift ratio SRSS (column)
EDP2: Residual drift ratio SRSS (column)

The Square Root of Sum of Squares (SRSS) values of the 2 horizontal components are used. The drift ratios are combined separately at each time step (to obtain SRSS).

EDP1 (Max drift ratio SRSS) is the maximum of the SRSS values of all time steps. EDP2 (Residual drift ratio SRSS) is the SRSS value at the last time step. The drift ratio is in percentage.

EDP3: Maximum longitudinal relative deck-end/abutment displacement (left)
EDP4: Maximum longitudinal relative deck-end/abutment displacement (right)

These two EDPs are intended to address the issue of abutment impact into the backwall, so they are defined as only the motion of the deck into the abutment. Maximum absolute values in the longitudinal direction are used.

EDP5: Maximum absolute bearing displacement (left abutment)
EDP6: Maximum absolute bearing displacement (right abutment)

These two EDPs are intended to address bearing damage whether or not an explicit representation of the bearings is included in the user-selected abutment model. Therefore, the EDP for the EDP is based on the relative displacements of the deck-end node to the abutment top node. The SRSS values of the resulting two relative horizontal displacements is used and both motion into the backwall and away from the backwall are considered.

EDP7: Residual vertical displacement (left abutment)
EDP8: Residual vertical displacement (right abutment)

This EDP is used to gage immediate repairs for rideability, and is not a measure of the permanent slumping of the embankment (for example). Therefore, the EDP is calculated as the vertical displacement of the abutment top node relative to the deck-end node. The residual value is used (value at the final time step).

The EDP quantities for all input motions can be accessed by clicking menu **Display** (Fig. 3) and then **EDP Quantities for All**. The window to display EDP quantities is shown in Fig. 113.

The EDP quantities are displayed against any of the 11 intensity measures. The EDP quantities for each input motion are displayed by bin of the motion (see legend in Fig. 113). When an IM is paired with an EDP and all the individual realizations are plotted, the result is typically termed a demand model, or probabilistic seismic demand model (PSDM).

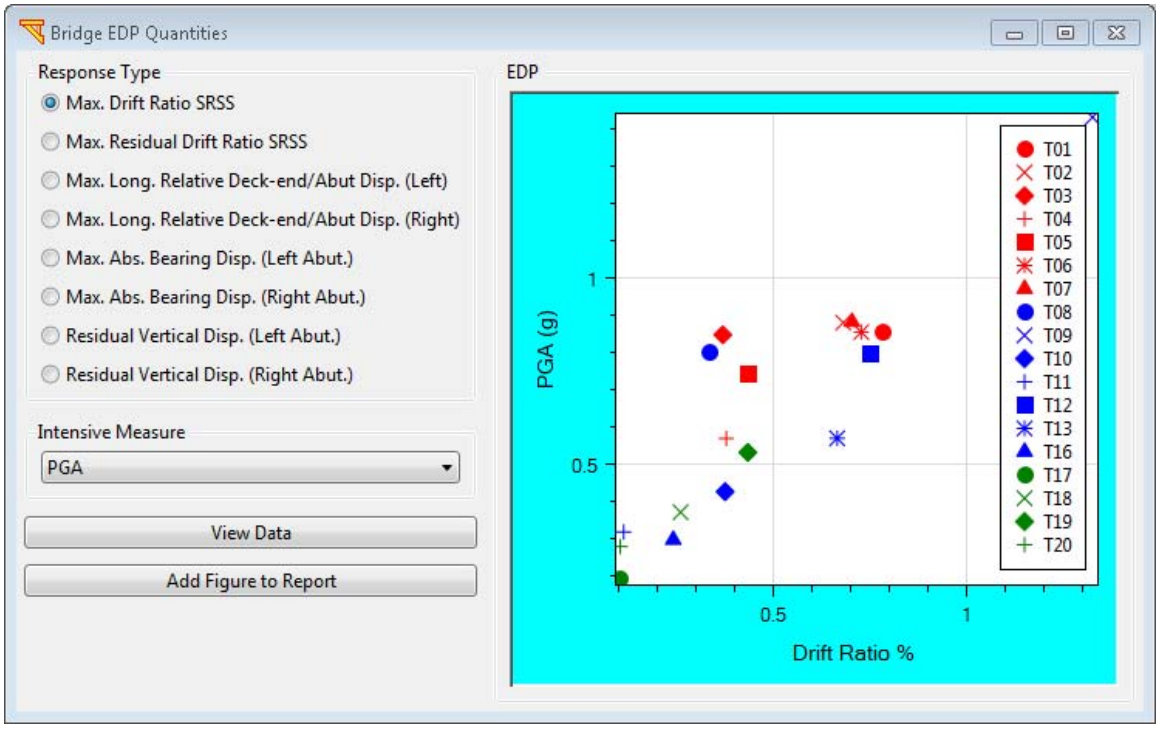


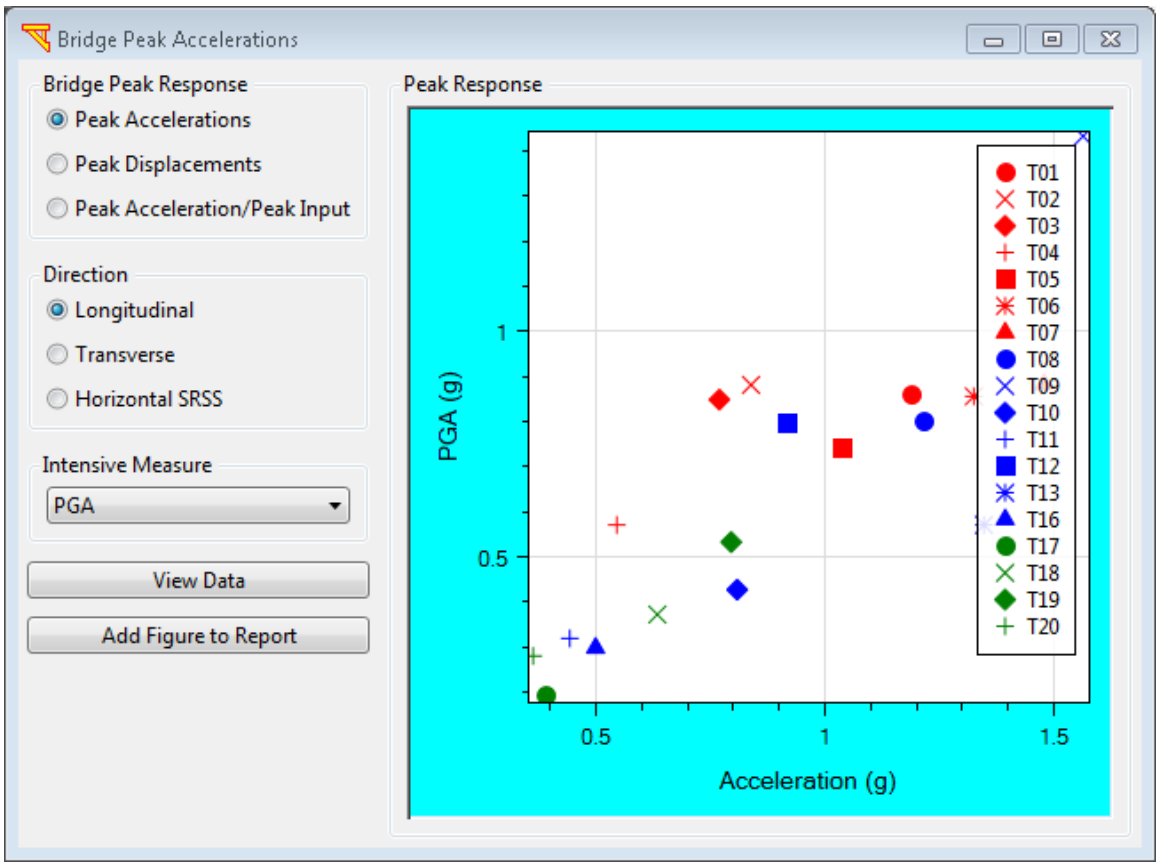
Fig. 113. EDP quantities for all motions

8.3.2 Bridge Peak Accelerations & Displacements for All Motions

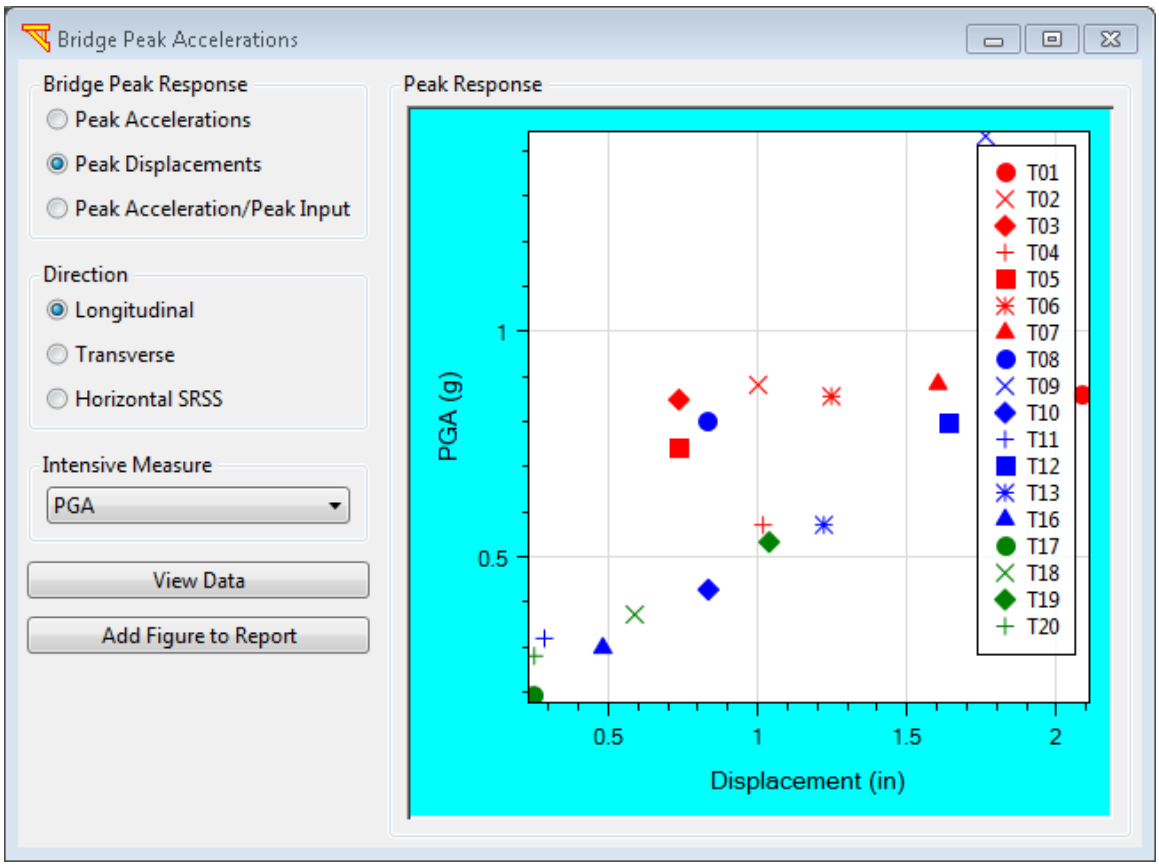
The bridge peak accelerations and displacements for all input motions can be accessed by clicking menu **Display** (Fig. 3) and then **Bridge Peak Accelerations& Displacements for All Motions**. The window to display the bridge peak accelerations for all motions is shown in Fig. 114. The responses are available in the longitudinal and transverse directions as well as for the SRSS of the 2 horizontal directions (Fig. 114).

The figures in this window include:

i) Maximum bridge acceleration
ii) Maximum bridge displacement
iii) Bridge peak acceleration / input peak acceleration


Bridge Peak Accelerations
Bridge Peak Response
Peak Accelerations
Peak Displacements
Peak Acceleration/Peak Input
Direction
Longitudinal
Transverse
Horizontal SRSS
Intensive Measure
PGA
View Data
Add Figure to Report
Peak Response
PGA (g)
Acceleration (g)
0.5
1
1.5
T01
T02
T03
T04
T05
T06
T07
T08
T09
T10
T11
T12
T13
T16
T17
T18
T19
T20

a)



b)

Fig. 114. Bridge peak accelerations for all motions: a) maximum bridge accelerations; b) maximum bridge displacements

8.3.3 Maximum Column & Abutment Forces for All Motions

The maximum column & abutment forces for all input motions can be accessed by clicking menu **Display** (Fig. 3) and then **Maximum Column & Abutment Forces for All Motions**. The window to display the maximum column & abutment forces for all motions is shown in Fig. 115. The responses are available in the longitudinal and transverse directions as well as for the SRSS of the 2 horizontal directions (Fig. 115).

The figures in this window include:

i) Maximum column shear forces
ii) Maximum column bending moments
iii) Maximum abutment forces (left abutment)
iv) Maximum abutment forces (right abutment)

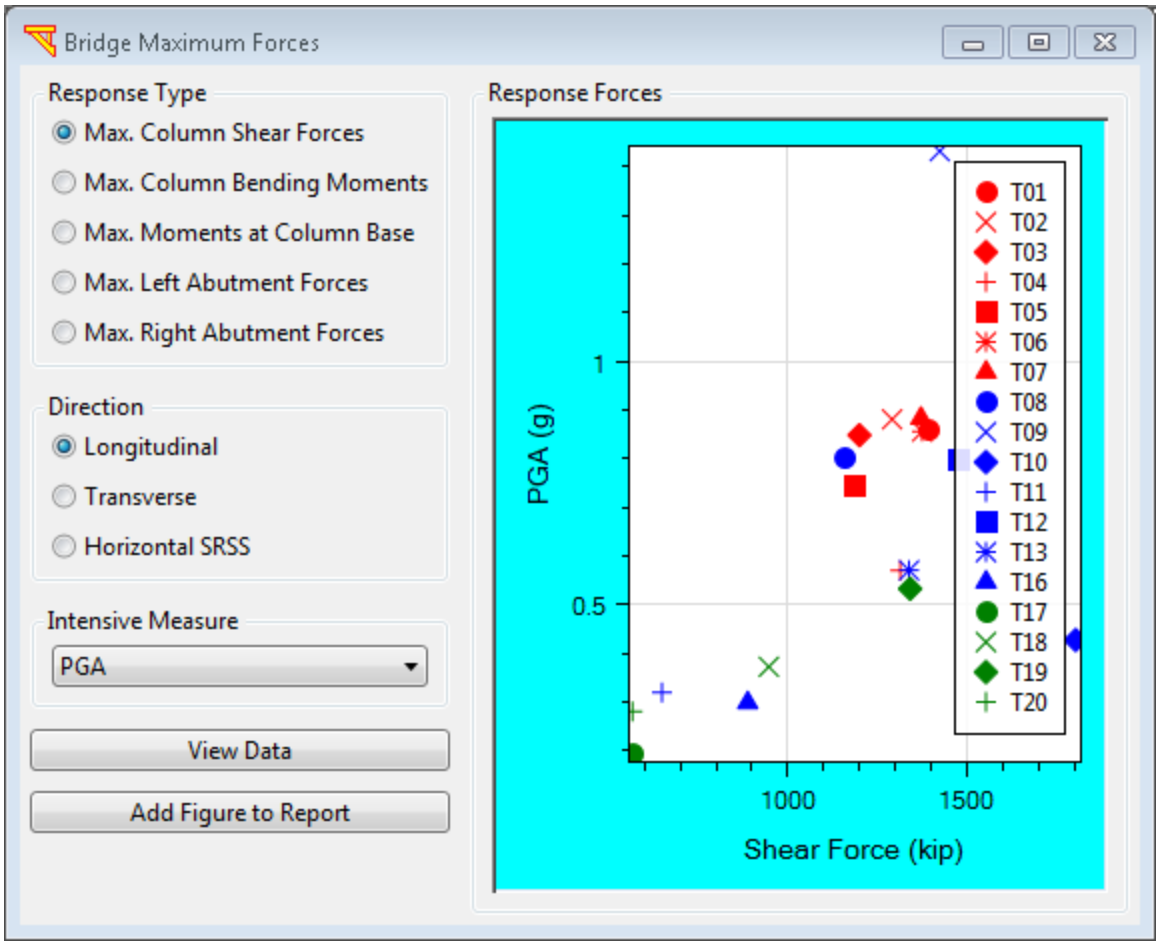


Fig. 115. Maximum column & abutment forces for all motions

9 Equivalent Static Analysis

Equivalent Static Analysis (ESA) option is available in **MSBridge** for the bridge longitudinal & transverse directions. The whole bridge system is employed in the bridge longitudinal ESA. And one single bent is employed in the bridge transverse ESA.

9.1 Bridge Longitudinal Direction

To conduct an Equivalent Static Analysis (ESA) for the bridge longitudinal direction, click **Longitudinal Direction** in the main window (Fig. 116). The elastic displacement demand output is shown in Fig. 117. The displacement demand output is available for the longitudinal components of the input motions.

To view the comparison of displacements from ESA and Time History Analysis (THA), click **Compare with THA**. The comparison result is shown in Fig. 118. However, the comparison is only available for ESA for the longitudinal components of the input motions (Fig. 117).

The procedure of the bridge longitudinal ESA is as follows:

1. Specify a load F of the total weight (see below for now to calculate the total weight), do pushover and get a displacement d
2. Calculate the Stiffness $K = F/d$
3. Calculate the Period $T = 2\pi$ sqrt(M/K)
4. From the spectral acceleration of the input motion, get Sa
5. Calculate $D_d = M*Sa/K$, and this is the elastic displacement demand.
6. Check the abutment displacement (D_d) compared to abutment yield displacement (D_y). If $D_d/D_y<2$, stop, D_d is the demand. If $D_d/D_y > 4$, set abutment spring to 0.1*its initial stiffness, recalculate the displacement demand (D_d). If $2 < D_d/D_y < 4$, linearly interpolate abutment stiffness between its full and 0.1 values and ratios of 2 and 4, then recalculate the displacement demand.

The whole bridge system is employed in the bridge longitudinal ESA. As such, the pushover load is applied at the bridge center along the bridge deck (longitudinal) direction.

The total weight is equal to the total deck weight plus ½ column weight. The deck weight should be distributed weight over span elements (or applied to nodes by tributary length). Column weight applied at top column node or if more than one element is used per column, distributed to column nodes by tributary length.

9.2 Bridge Transverse Direction

To conduct an ESA for the bridge transverse direction, click Transverse Direction in the main window (Fig. 116). The output is shown in Fig. 119.

Only one single bent is employed in the bridge transverse ESA. As such, the pushover load is applied at the bent cap center along the bent cap direction (bridge transverse direction).

The total weight is equal to the deck weight of left half span and right half span for the bent plus ½ column weight.

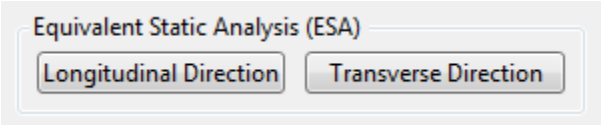
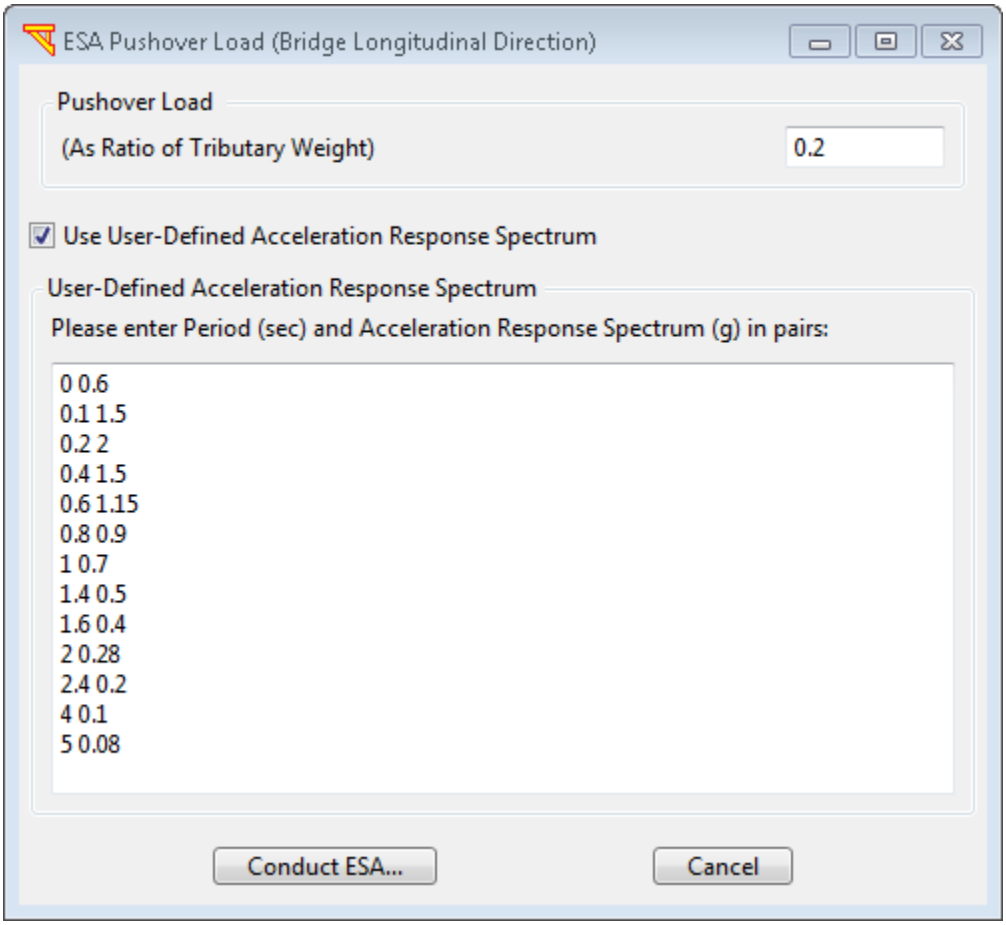
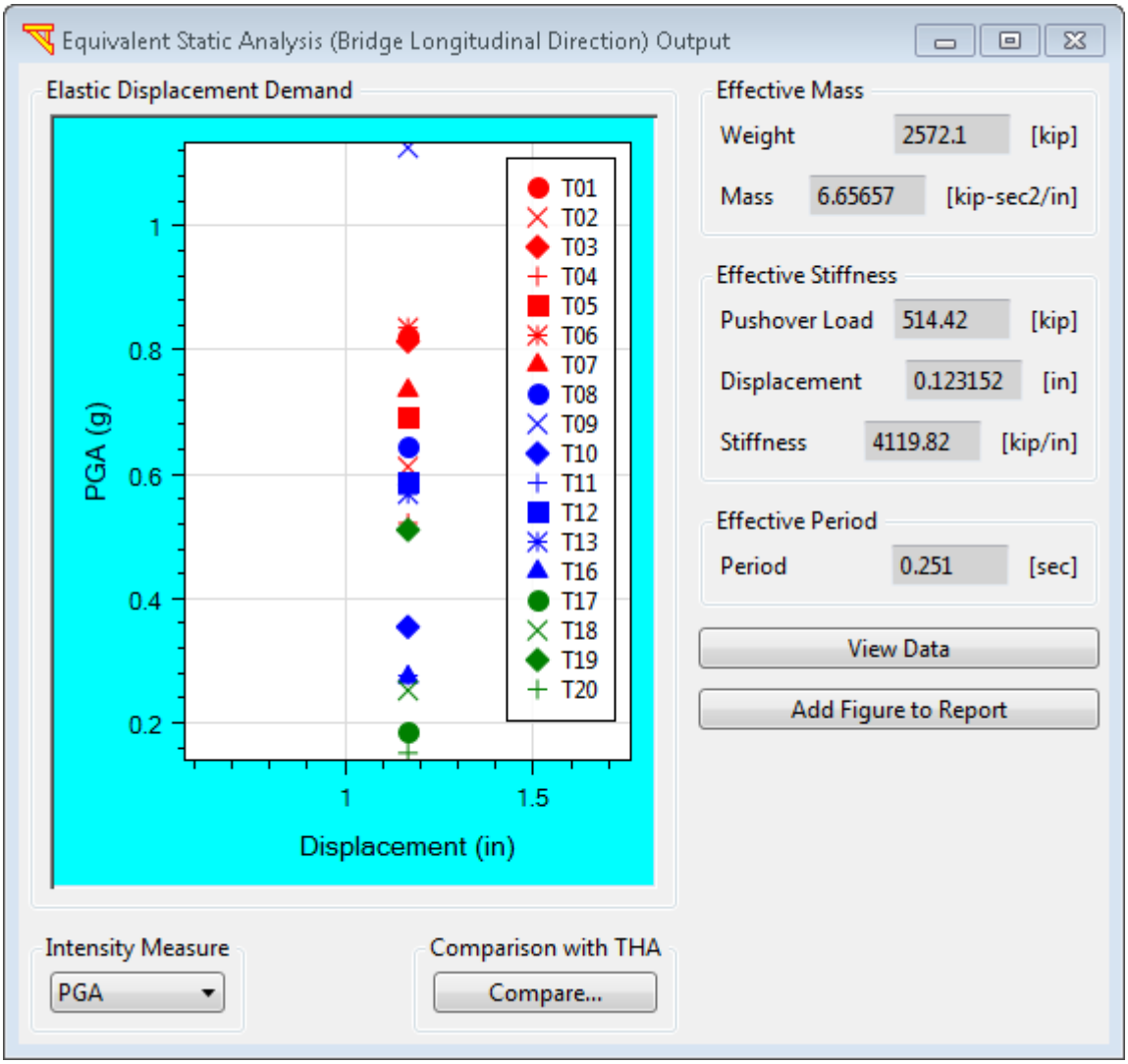


Fig. 116. Equivalent Static Analysis for the bridge longitudinal & transverse directions

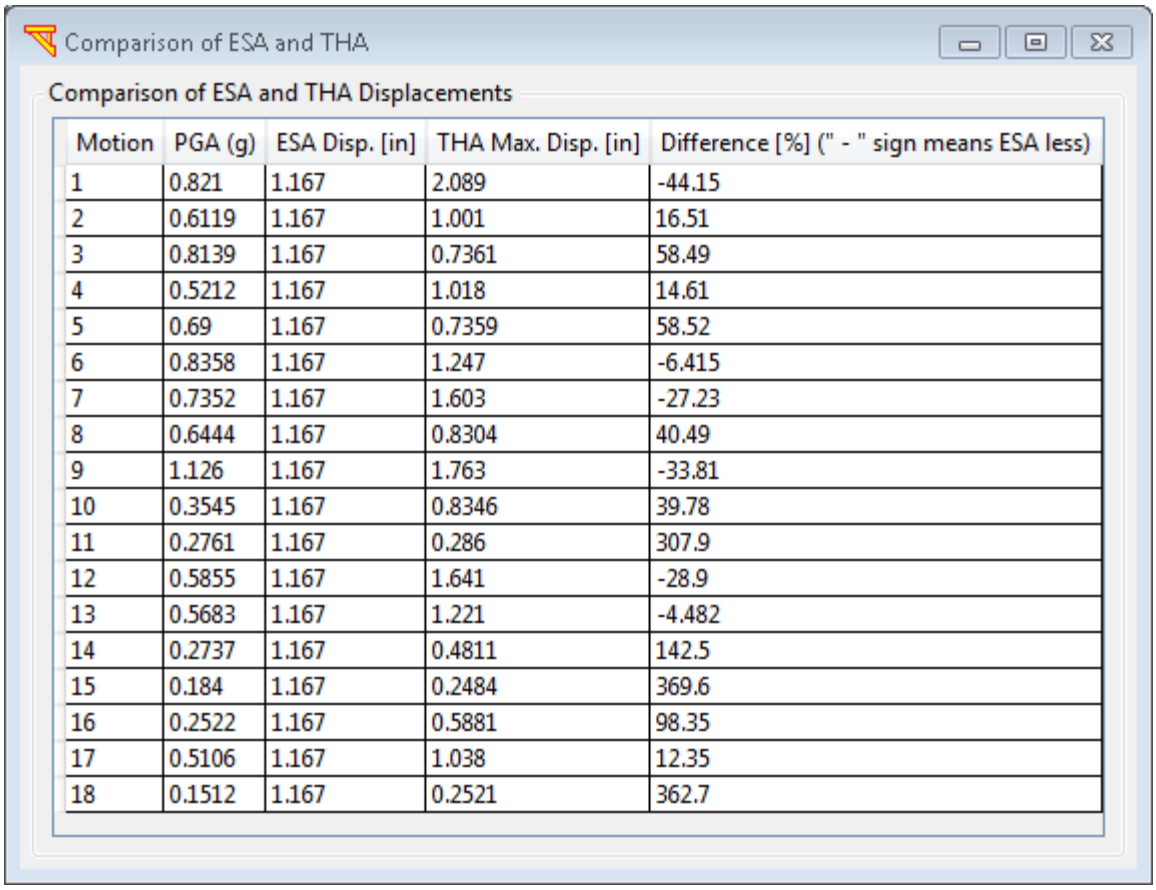


a)



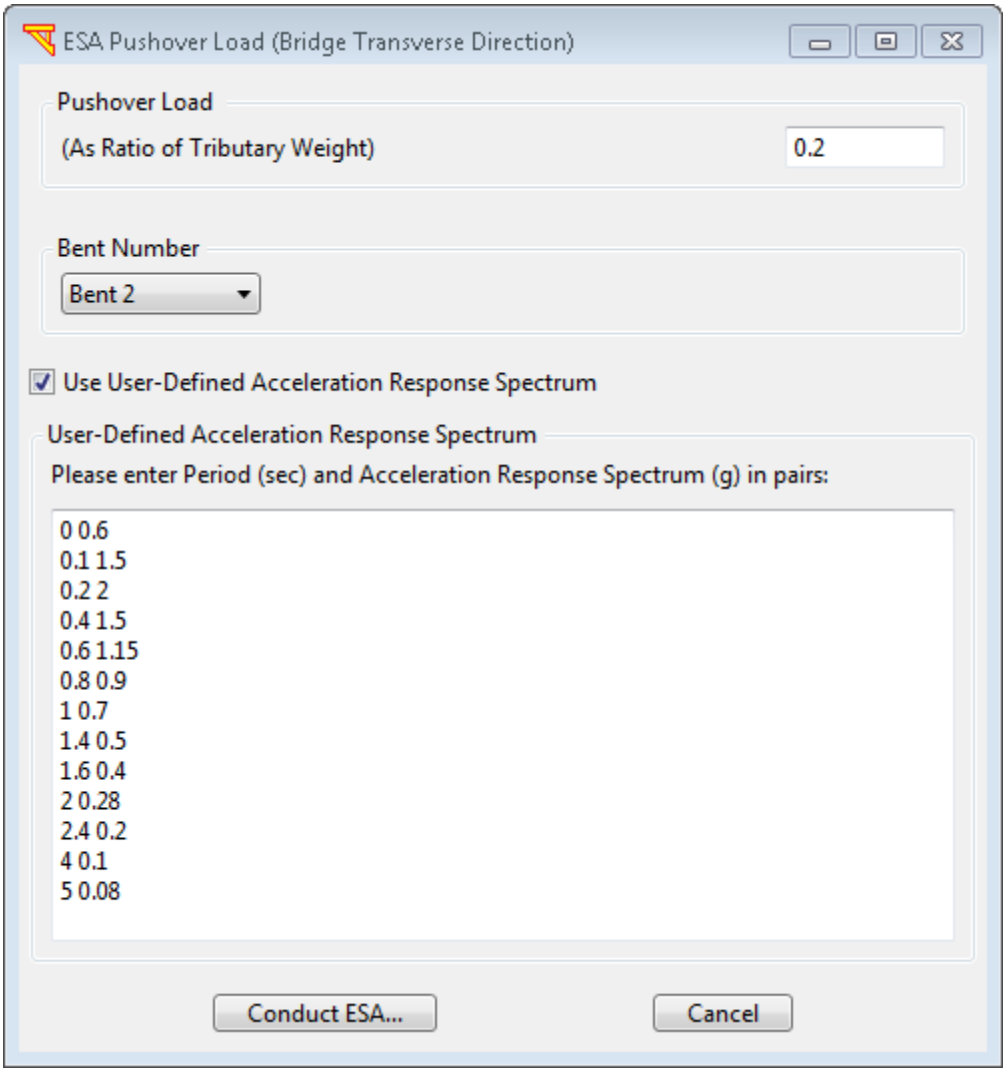
b)

Fig. 117. Sample output of ESA for the bridge longitudinal direction: a) pushover load; b) elastic displacement demand

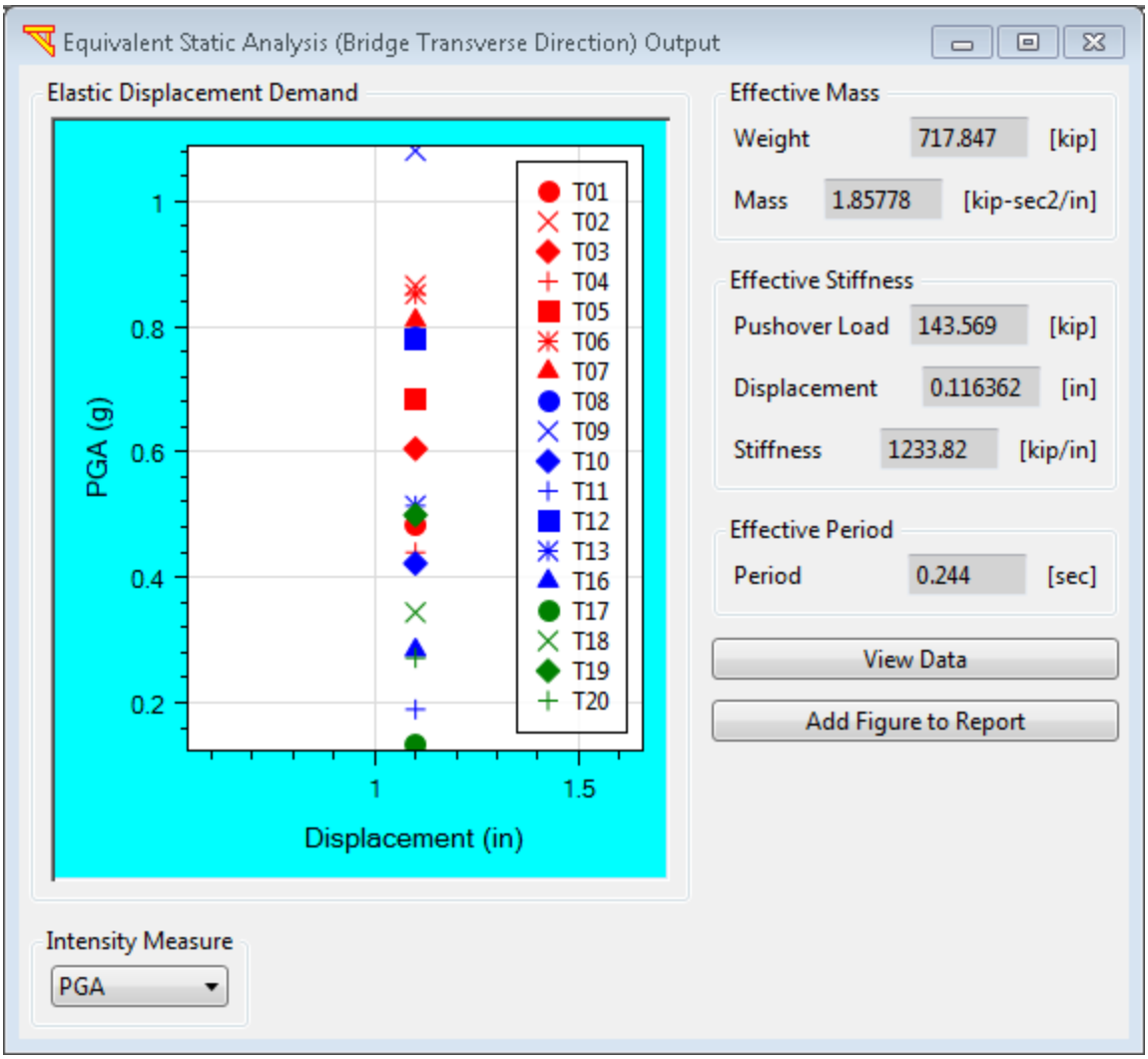


Motion	PGA (g)	ESA Disp. [in]	THA Max. Disp. [in]	Difference [%] (" - " sign means ESA less)
1	0.821	1.167	2.089	-44.15
2	0.6119	1.167	1.001	16.51
3	0.8139	1.167	0.7361	58.49
4	0.5212	1.167	1.018	14.61
5	0.69	1.167	0.7359	58.52
6	0.8358	1.167	1.247	-6.415
7	0.7352	1.167	1.603	-27.23
8	0.6444	1.167	0.8304	40.49
9	1.126	1.167	1.763	-33.81
10	0.3545	1.167	0.8346	39.78
11	0.2761	1.167	0.286	307.9
12	0.5855	1.167	1.641	-28.9
13	0.5683	1.167	1.221	-4.482
14	0.2737	1.167	0.4811	142.5
15	0.184	1.167	0.2484	369.6
16	0.2522	1.167	0.5881	98.35
17	0.5106	1.167	1.038	12.35
18	0.1512	1.167	0.2521	362.7

Fig. 118. Comparison of displacements from ESA and THA


ESA Pushover Load (Bridge Transverse Direction)
Pushover Load
(As Ratio of Tributary Weight)
0.2
Bent Number
Bent 2
Use User-Defined Acceleration Response Spectrum
User-Defined Acceleration Response Spectrum
Please enter Period (sec) and Acceleration Response Spectrum (g) in pairs:
0 0.6
0.1 1.5
0.2 2
0.4 1.5
0.6 1.15
0.8 0.9
1 0.7
1.4 0.5
1.6 0.4
2 0.28
2.4 0.2
4 0.1
5 0.08
Conduct ESA...
Cancel

a)



b)

Fig. 119. Output of ESA for the bridge transverse direction: a) pushover load and bent number; b) elastic displacement demand

Appendix A Calculation of Steel and Concrete Material Properties

Steel Bars

By default, the Steel02 material is used to simulate steel bars. The format of the Steel02 command is as follows (McKenna et al. 2010):

uniaxialMaterial Steel02 $matTag $fy $E0 $b $R0 $cR1 $cR2

Where $fy is the steel yield strength, $E0 is Young's modulus of steel, and $b is the strain-hardening ratio (ratio between post-yield tangent and initial elastic tangent), $R0, $cR1 and $cR2 are parameters to control the transition from elastic to plastic branches.

The number of longitudinal bars is calculated as follows:

$$\#bars = \frac{\rho_s A_c}{A_b} \tag{5}$$

Where ρ_s is the longitudinal steel percentage, A_c the column cross-section area, A_b is the cross-section area of the steel bar.

If the number of longitudinal bars is known, the longitudinal steel percentage (reinforcement ratio) can be calculated:

$$\rho_s = \frac{A_s}{A_c} \tag{6}$$

Where A_s is the area of longitudinal steel, which is equal to the area of each bar times the number of bars. For example, the diameter of a #18 bar is 2.257 inches, so area is 4 in^2. If there are 10 bars in a 36 inch diameter circular column, then

$$\rho_s = \frac{10(4)}{\frac{\pi}{4}36^2} = 0.039$$

or 3.9%.

The transverse steel percentage (reinforcement ratio) for a spirally confined circular column, currently the only type of column supported in the interface, is

$$\rho_t = \frac{\pi(d_{db}{}^2)}{s(d_{cc})} \tag{7}$$

Where d_{bt} is the diameter of the transverse spiral (always smaller than the diameter of the longitudinal bars). The spacing between transverse bars is *s*. The diameter of the confined core is d_{cc} which is the gross diameter minus twice the cover and minus the diameter of the transverse bars (see Eq. 10). So for a #5 spiral spaced at 3 inches on center in the same column mentioned above.

$$\rho_t = \frac{\pi(\frac{5}{8})^2}{3(36-2(2)-\frac{5}{8})} = 0.013 \tag{8}$$

or 1.3%.

Currently the transverse reinforcement does affect the shear response (through changes in the uniaxial constitutive model for the concrete core). However, the columns are modeled considering only flexurally dominated response (i.e., there is no accounting for shear flexibility or shear degradation directly). Additional relevant details on the parameters used in both the Cover and Core Concrete are included below.

Cover concrete

The Concrete02 material is used to simulate the concrete (for both cover and core). The format of the Concrete02 command is as follows:

uniaxialMaterial Concrete02 $matTag $fpc $epsc0 $fpcu $epsu $lambda $ft $Ets

Where $fpc is the concrete compressive strength, $epsc0 is the concrete strain at maximum strength, $fpcu is the concrete crushing strength, $epsu is the concrete strain at crushing strength (all of the above values are entered as negative), $lambda is the ratio between unloading slope at $epsu and initial slope, $ft is the tensile strength, and $Ets is tension softening stiffness (absolute value) (slope of the linear tension softening branch).

For cover concrete, $fpc is equal to the concrete unconfined strength**Error! Reference source not found.**, $epsc0 = 0.002, $fpcu = 0.0, $epscu = 0.006, $lambda = 0.1, $ft = (0.14)$fpc, and $Ets = $ft / $epsc0.

Core concrete

i) For core concrete of circular column cross sections according to the Mander model, the procedure to calculate the confined concrete strength $fpc(= f_{cc}) is as follows:

$$f_{cc} = f_c^{'}(-1.254 + 2.254\sqrt{1 + 7.94\frac{f_e^{'}}{f_c^{'}}} - 2\frac{f_e^{'}}{f_c^{'}}) \quad (9)$$

Where $f_c^{'}$ is the unconfined compressive strength and $f_e^{'}$ can be obtained from the following equation:

$$f_e^{'} = \frac{1}{2} K_e \rho_t f_y \quad (10)$$

Where f_y is the steel yield strength, ρ_t is the transverse steel percentage, and K_e can be obtained from the following equation for spirally confined circular columns:

$$K_e = \frac{(1 - \frac{S'}{2d_{cc}})^2}{(1 - \rho_{cc})} \quad (11)$$

Where:

$$\rho_{cc} = \frac{A_s}{A_{cc}} \quad (12)$$

An assumed value of the area of the confined core is used for default values. This area should be modified based on the expected compressive block in the column during lateral loading.

$$A_{cc} = \frac{\pi (d_{cc})^2}{4} \quad (13)$$

$$S^{'} = \frac{\pi d_{bt}^{\ 2}}{\rho_t d_{cc}} \quad (14)$$

Where d_{bt} is the transverse bar diameter

$$d_{cc} = D_L - 2c - d_{bt} \quad (15)$$

Where c is the clear cover (c = 1.5")

ii) $epsc0

$$epsc0 = \frac{2 f_{cc}}{E_c} \quad (16)$$

Where:

$$Ec = 0.043w^{1.5}\sqrt{f_c^{'}} \tag{17}$$

Where *w* is the concrete unit weight (unit: kg/m^3)

iii) $epsu (= *epscu*)

$$epscu = 0.004 + \varepsilon_s \frac{f_y}{f_c^{'}} \rho_t \tag{18}$$

Where ε_s is the ultimate steel strain ($\varepsilon_s = 0.12$)

iv) $fpcu (= f_{cu})

$$f_{cu} = \frac{f_{cc}(epscu)}{(epsc)} \left(\frac{(epscr)}{(epscr) - 1 + \left(\frac{(epscu)}{(epsc)}\right)^{(epscr)}} \right) \tag{19}$$

Where:

$$epsc = (epsc0)\left(1 + 5\left(\frac{f_{cc}}{f_c^{'}} - 1\right)\right) \tag{20}$$

$$epscr = \frac{E_c}{E_c - \frac{f_{cc}}{(epsc)}} \tag{21}$$

Notes:

1. The information above is specific to the Steel02 and Concrete02 models of the Fiber section. Other options include (Fig. 27), Steel01 and Concrete01 (for more information please see the OpenSees documentation), and Elastic properties for the fibers. These options can be activated by clicking on the default Steel02 or Concrete02 sections (Fig. 27) and changing these options.

2. A different property may be specified for the Column below grade (for instance to roughly represent a large pile group as a large single column). If this option is selected

(Fig. 6), the column below grade will have linear properties as specified by its diameter and Young's Modulus).

3. All the equations presented in this Appendix are based on the Mander model for spiral-reinforced circular concrete columns. The user may want to use their own constitutive model or parameters. In this case, the values of these parameter can be defined directly in Fig. 27.

Appendix B How to Incorporate User-defined Motions

1) Directory Structure of a Motion Set

To conduct a base input acceleration analysis, input motions must be defined (Fig. 91). The window to define the input motions is shown in Fig. 92. Click **Browse** to select a motion set (Fig. 120). Click on the motion set name (e.g., **Motions**) and then click on **OK** to choose this motion set (Fig. 120).

In MSBridge, the input motions are organized in a format that the program can read. Specially, the input ground motions are sorted into bins. Fig. 121 shows the directory structure of a motion set named **Motions**. The second level directories are bins (e.g., T01; see Fig. 120 and Fig. 121). The third level directories are earthquake names (e.g., there is earthquake NORTHRIDGE; see Fig. 121). And the fourth level directories are the input motion names (e.g., there is 1 input motion under earthquake NORTHRIDGE: RRS; see Fig. 121).

Each motion is composed of 3 perpendicular acceleration time history components (2 laterals and one vertical). As shown in Fig. 121, each motion folder contains 6 files categorized into 2 file types: the DATA files contain the time history (acceleration unit in g) of a component and the INFO files contain the characteristics of the corresponding component. Fig. 122 and Fig. 123 displays sample INFO & DATA files. Naming of these files usually has to follow the format below: Input motion name + angle (or "-UP" or "-DWN" for vertical component) + ".AT2" + ".data" (or ".info"). However, the following format is also allowed: Input motion name + "-E" or "-W" for horizontal components (or "-V" for vertical component).

Note that the filenames with the smaller angle will be used for the longitudinal direction and the other one (with the larger angle) will be used for the transverse direction (also, the filenames containing "-E" will be used for the longitudinal direction and the other one (containing "-W") will be used for the transverse direction.

The first 2 lines of each INFO file must follow the style of the example below:
{Data points NPTS} {996}
{Sampling period DT (sec)} {0.020000}

Where 996 and 0.02 are the number of data points, and the time step, respectively, of an input motion component.

2) Steps to Create an Input Motion

Based on the above description for the directory structure of a motion set, one can easily create an input motion (Fig. 124):

Step 1: create a folder and rename to your motion set name (e.g. MotionSet1; see Fig. 124).

Step 2: create a folder under the motion set folder and rename to your bin name (e.g., bin1).

Step 3: create a folder under the bin folder and rename to your earthquake name (e.g., Quake1).

Step 4: create a folder under the earthquake name and rename to your input motion name (e.g. MOTION1).

Step 5: create the 6 files (3 INFO files and 3 DATA files) for this input motion (Fig. 124).

Note: If you download the input motion files from the PEER NGA Database, there is no need to re-format the data into one column as shown in Fig. 123. Just copy the data points into the corresponding DATA files. And then make the INFO files containing the number of data points and the sampling period DT (2 lines) according to the header information.

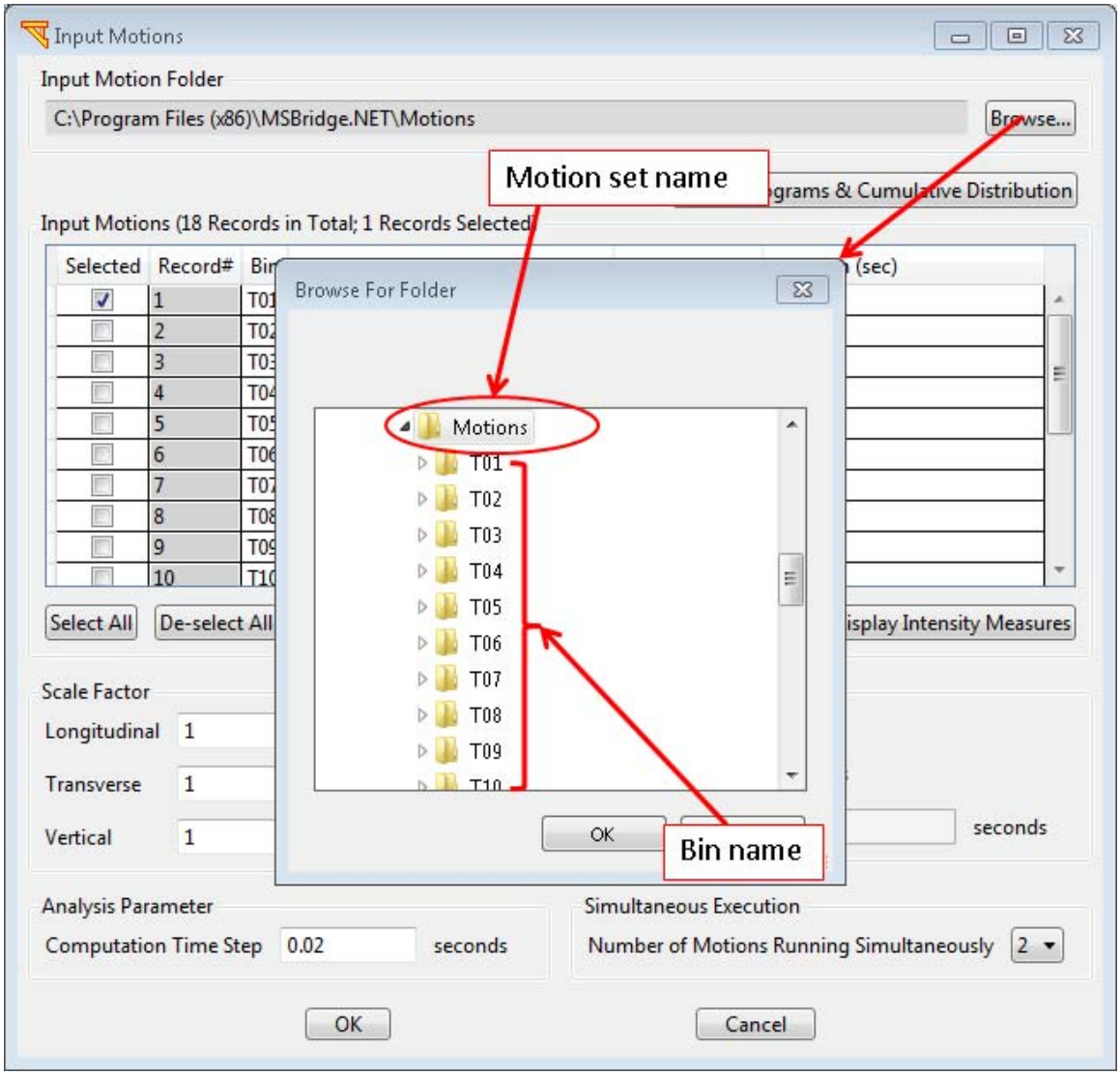


Fig. 120. Choosing a motion set

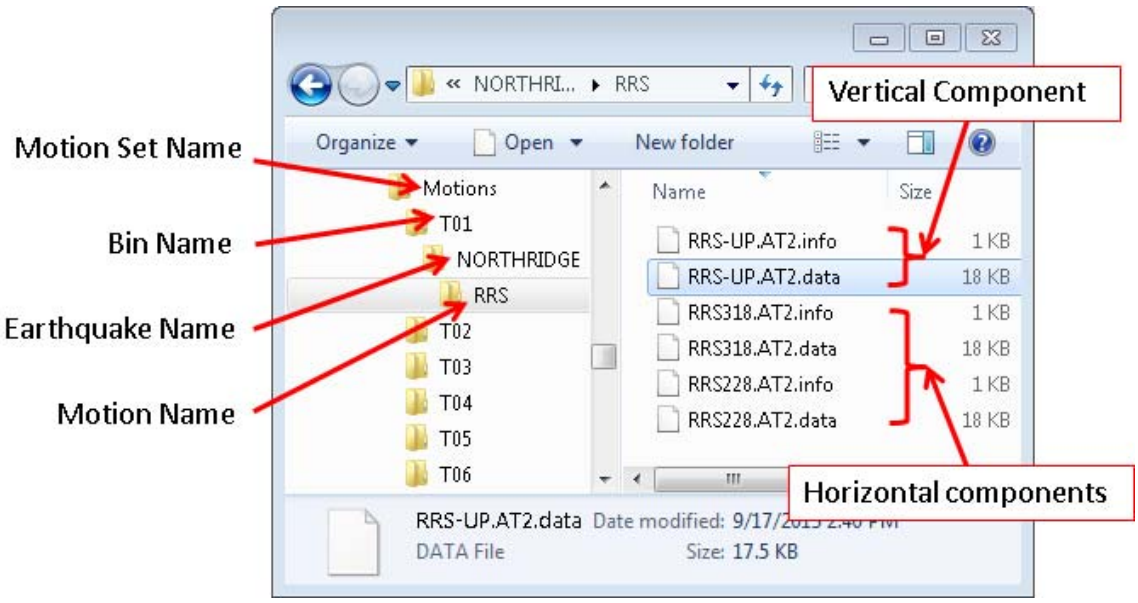


Fig. 121. Directory structure of a motion set

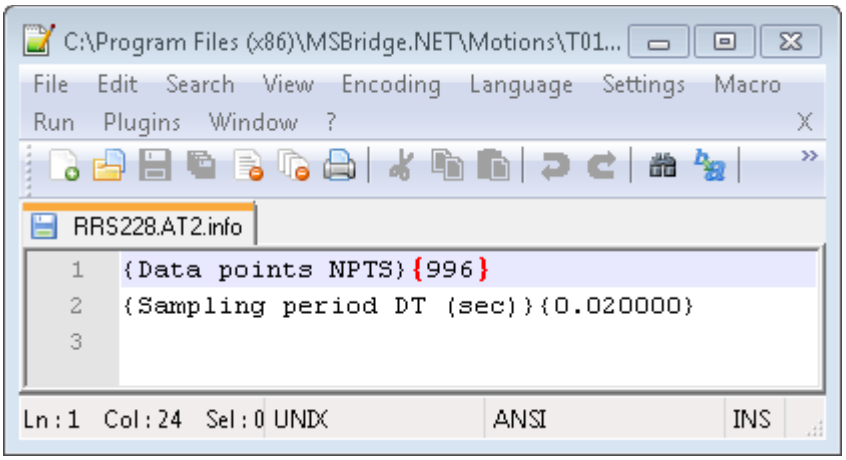

C:\Program Files (x86)\MSBridge.NET\Motions\T01...
File Edit Search View Encoding Language Settings Macro
Run Plugins Window ?
RRS228.AT2.info
1 {Data points NPTS}{996}
2 {Sampling period DT (sec)}{0.020000}
3
Ln : 1 Col : 24 Sel : 0 UNIX ANSI INS

Fig. 122. Sample .info file

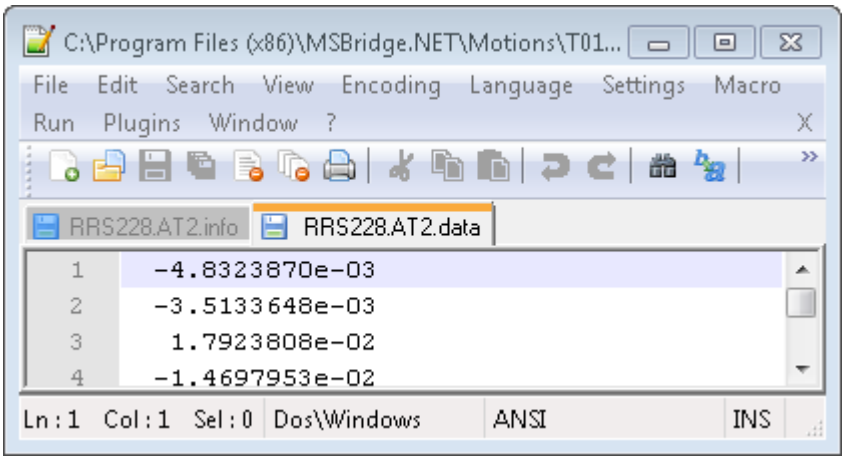


Fig. 123. Sample .data file

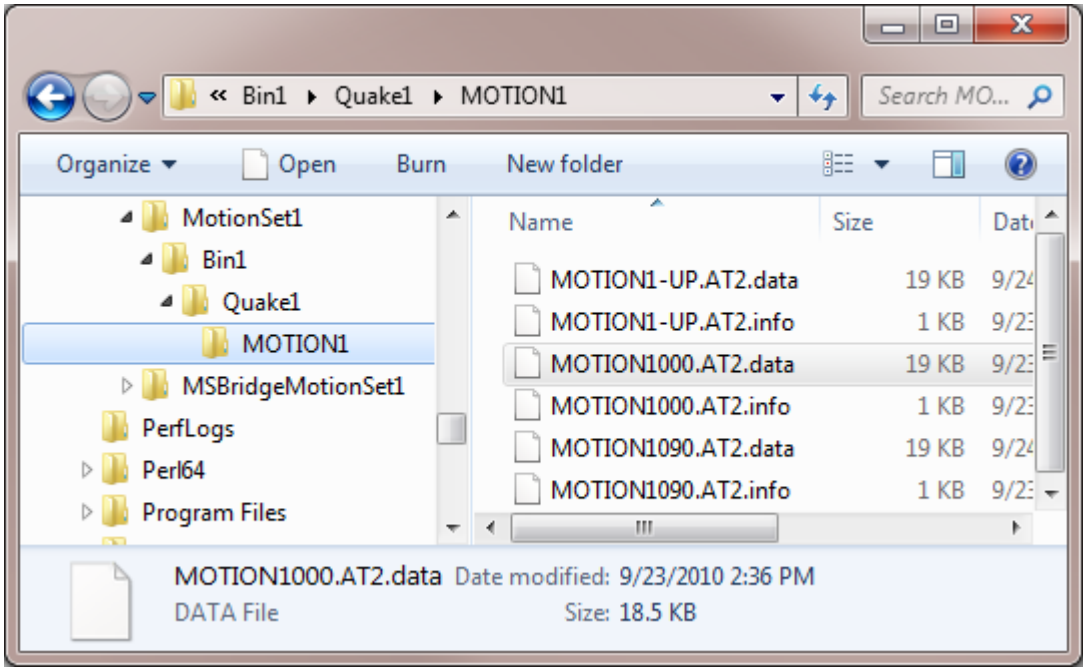


Fig. 124. Example of user-defined motion

Appendix C Comparison with SAP2000 for Representative OB Configurations

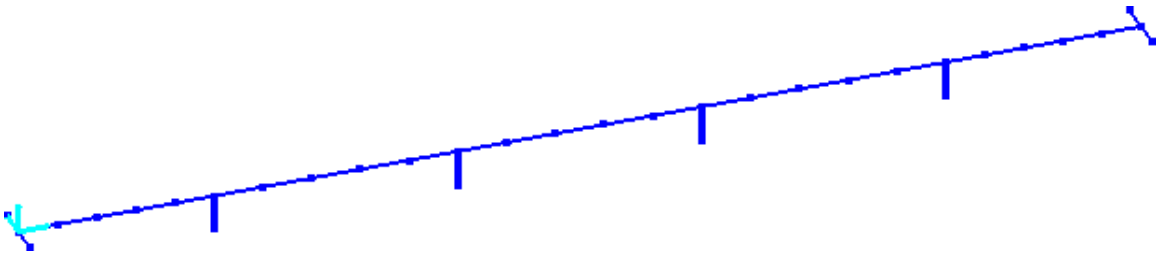
A large portion of bridges in the current California bridge inventory share similar construction characteristics, especially those owned and maintained by the California Department of Transportation (CalTrans) (Mackie and Stojadinovic, 2007). Eleven bridge configurations were selected by Ketchum et al (2004) as representative of typical statewide bridge construction in California. These bridge configurations are listed in Table 7.

Table 7. Typical Bridge Configurations in California (After Ketchum et al 2004)

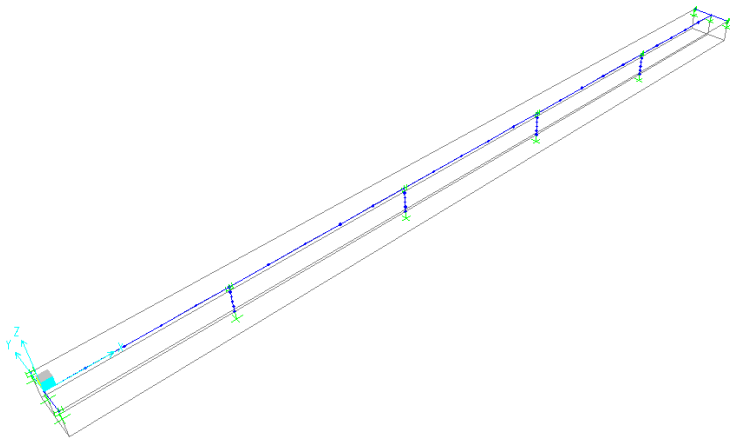
Bridge Type	Span Arrangement	Geometry	Bent Columns	Column Height	Column Height	Deck Width
1	120'+150'+150'+150'+120'	Straight	1	22'	39'	6'
2	120'+150'+150'+150'+120'	Straight	3	22'	68'	6'
3	80'+100'+100'+100'+80'	Straight	1	22'	39'	4'
4	80'+100'+100'+100'+80'	Straight	3	22'	68'	4'
5	80'+100'+100'+100'+80'	Straight	1	22'	39'	5'-2"
6	80'+100'+100'+100'+80'	Straight	3	22'	68'	5'-2"
7	120'+120'	Straight	1	22'	39'	6'-2"
8	120'+120'	Straight	3	22'	68'	6'-2"
9	120'+150'+150'+150'+120'	1000' radius	1	22'	27'	6'
10	80'+100'+100'+100'+80'	30 skew	3	22'	68'	4'
11	120'+150'+150'+150'+120'	Straight	1	50'	39'	6'

The above models were built (without much effort) in **MSBridge** (Linear columns, Roller abutment model and Rigid-base were assumed; default values were used for other bridge parameters). The SAP2000 models (.s2k files) were obtained by clicking Menu File and then Export, SAP2000 .s2k Text File.

Linear analyses of monotonic pushover show both **MSBridge** and SAP2000 gave the identical results for all of the 11 bridge configurations shown in Table 7. For example, Fig. 125 shows the models built in **MSBridge** and SAP2000 for Bridge Type 1. Table 8 shows the displacement of the deck at each bent under the pushover load of 2000 kips applied at the deck center along the longitudinal and transverse directions. Fig. 126 and Table 9 show the comparison for Bridge Type 2. Fig. 127 and Table 10 show the comparison for Bridge Type 9. Fig. 128 and Table 11 show the comparison for a skewed bridge case (Bridge Type 10). Both **MSBridge** and SAP2000 essentially gave the same result.



a)

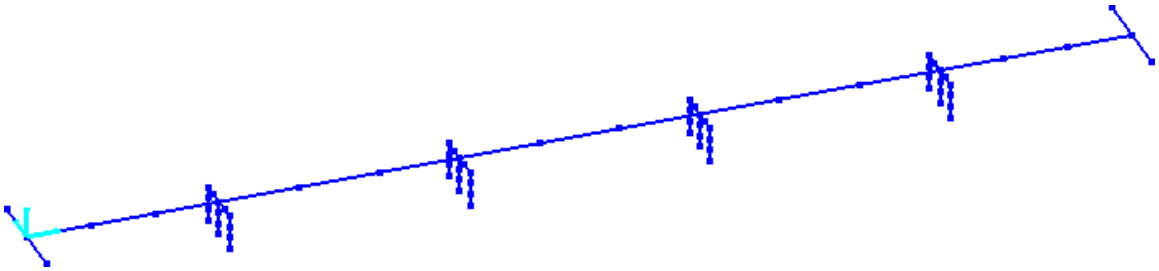


b)

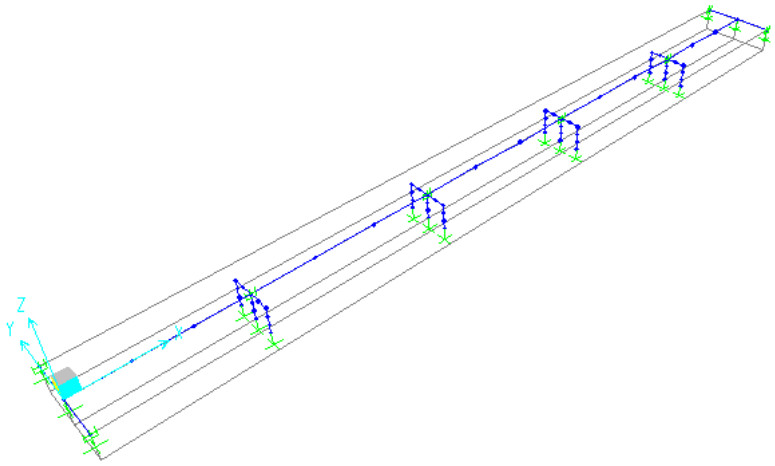
Fig. 125. Bridge Type 1 model: a) **MSBridge**; b) SAP2000

Table 8. Displacement (unit: inch) of Bridge Type 1 under pushover (load of 2000 kips applied at deck center along both the longitudinal and transverse directions)

	Middle Column in Bent	Longitudinal Displacement	Transverse Displacement	Vertical Displacement
SAP2000	1	0.71745	0.68199	-0.05127
	2	0.73926	2.38939	-0.05235
	3	0.73025	2.15946	-0.05204
	4	0.70799	0.50487	-0.05278
MSBridge	1	0.71724	0.68326	-0.0513
	2	0.73941	2.39682	-0.0524
	3	0.73064	2.15976	-0.0520
	4	0.70873	0.50778	-0.0528
Difference	1	0%	0%	0%
	2	0%	0%	0%
	3	0%	0%	0%
	4	0%	1%	0%



a)

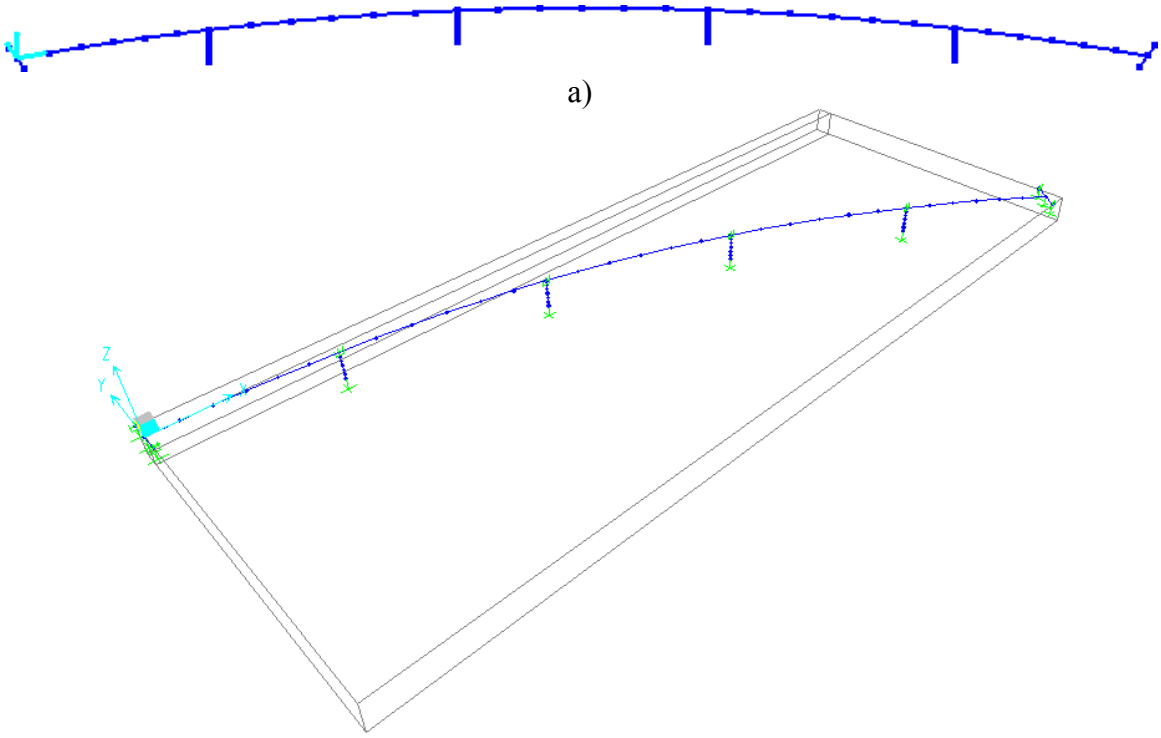


b)

Fig. 126. Bridge Type 2 model: a) **MSBridge**; b) SAP2000

Table 9. Displacement (unit: inch) of Bridge Type 2 under pushover (load of 2000 kips applied at deck center along both the longitudinal and transverse directions)

	Middle Column in Bent	Longitudinal Displacement	Transverse Displacement	Vertical Displacement
SAP2000	1	0.3051	0.117	-0.0728
	2	0.3194	0.4988	-0.074
	3	0.3094	0.4082	-0.0737
	4	0.2946	0.0304	-0.0742
MSBridge	1	0.3053	0.1159	-0.073102
	2	0.3201	0.5099	-0.074236
	3	0.3104	0.4097	-0.074008
	4	0.2961	0.03116	-0.074469
Difference	1	0%	1%	0%
	2	0%	2%	0%
	3	0%	0%	0%
	4	1%	3%	0%



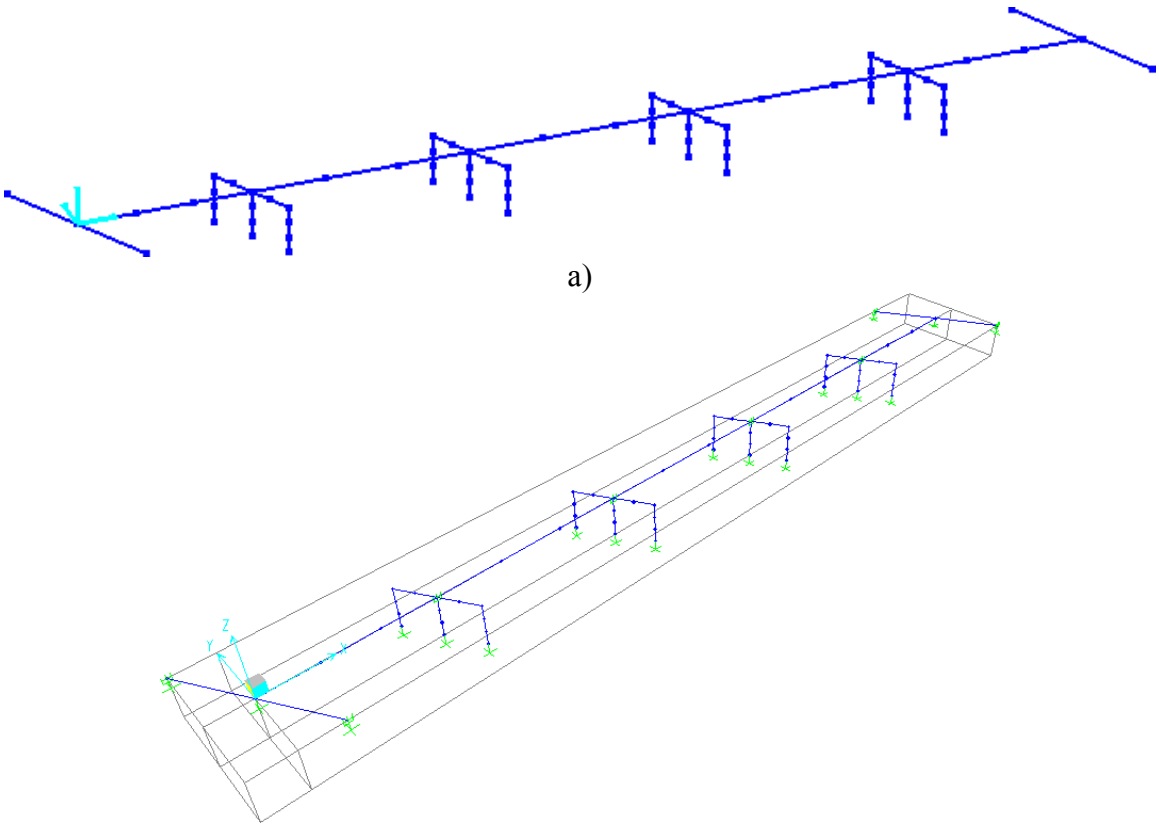
a)

b)

Fig. 127. Bridge Type 9 model: a) **MSBridge**; b) SAP2000

Table 10. Displacement (unit: inch) of Bridge Type 9 under pushover (load of 1000 kips applied at deck center along both the longitudinal and transverse directions)

	Middle Column in Bent	Longitudinal Displacement	Transverse Displacement	Vertical Displacement
SAP2000	1	0.4862	0.1208	-0.0410
	2	0.7547	1.3601	-0.0423
	3	0.7268	1.3913	-0.0422
	4	0.0953	0.2112	-0.0415
MSBridge	1	0.4881	0.1240	-0.0410
	2	0.7571	1.3630	-0.0423
	3	0.7266	1.3950	-0.0422
	4	0.0956	0.2149	-0.0415
Difference	1	0%	3%	0%
	2	0%	0%	0%
	3	0%	0%	0%
	4	0%	2%	0%



a)

b)

Fig. 128. Bridge Type 10 model: a) **MSBridge**; b) SAP2000

Table 11. Displacement (unit: inch) of Bridge Type 10 under pushover (load of 2000 kips applied at deck center along both the longitudinal and transverse directions)

	Middle Column in Bent	Longitudinal Displacement	Transverse Displacement	Vertical Displacement
SAP2000	1	0.3910	0.3054	-0.0456
	2	0.4018	0.4871	-0.0472
	3	0.3935	0.4210	-0.0476
	4	0.3811	0.1755	-0.0474
MSBridge	1	0.3931	0.2917	-0.0457
	2	0.4046	0.4991	-0.0472
	3	0.3970	0.4309	-0.0477
	4	0.3852	0.1853	-0.0475
Difference	1	1%	5%	0%
	2	1%	2%	0%
	3	1%	2%	0%
	4	1%	5%	0%

References

Aviram, A., Mackie, K.R., and Stojadinovic, B. (2008). Effect of Abutment Modeling on the Seismic Response of Bridge Structures, Earthquake Engineering and Engineering Vibration 7(4), 395-402.

Berry, M.P., and Eberhard, M.O. (2007). Performance Modeling Strategies for Modern Reinforced Concrete Bridge Columns, Report No. 2007/07, Pacific Earthquake Engineering Research Center, University of California, Berkeley, California.

Kotsoglu, A., and Pantazopoulou, S. (2006). Modeling of Embankment Flexibility and Soil-structure Interaction in Integral Bridges, Proceedings of First European Conference on Earthquake Engineering and Seismology, September 3-8, Geneva, Switzerland.

Caltrans SDC, 2004. Caltrans Seismic Design Criteria, Version 1.3, California Department of Transportation, Sacramento, California.

Caltrans SDC, 2006. Caltrans Seismic Design Criteria, Version 1.4, California Department of Transportation, Sacramento, California.

Caltrans SDC, 2010. Caltrans Seismic Design Criteria, Version 1.6, California Department of Transportation, Sacramento, California.

Elgamal, A., Yang, Z., Parra, E., and Ragheb, A. 2003. Modeling of cyclic mobility in saturated cohesionless soils. International Journal of Plasticity, 19(6), 883-905.

Elgamal, A., and Lu, J. 2009. A Framework for 3D finite element analysis of lateral pile system response, Proceedings of the 2009 International Foundation Congress and Equipment Expo, Contemporary Topics in In Situ Testing, Analysis, and Reliability of Foundations, ASCE GSP 186, M. Iskander, D. F. Laefer, and M. H. Hussein, Editors, Orlando, Florida, March 15-19, pp. 616-623.

Elgamal, A., Jinchi Lu, J., and Forcellini, D. 2009a. Mitigation of liquefaction-induced lateral deformation in a sloping stratum: 3D numerical simulation, Journal of geotechnical and geoenvironmental engineering, ASCE, Vol. 135, No. 11, November , 1672-1682.

Elgamal, A., Lu, J., Yang, Z. and Shantz, T. 2009b. Scenario-focused three-dimensional computational modeling in geomechanics, Alexandria, Egypt, October 3-5, 4 iYGEC'09 - 4th International Young Geotechnical Engineers' Conference, 2 - 6 October, ISSMGE.

Elgamal, A. (2010). Calibrated 3D computational modeling of soil-structure systems and liquefaction scenarios, Proc. Fifth Intl. Conf. on Recent Advances in Geotechnical Earthquake Engineering and Soil Dynamics, May 24-29, San Diego, CA.

Lu J., 2006. Parallel finite element modeling of earthquake site response and liquefaction, PhD Thesis, Department of Structural Engineering, University of California, San Diego, La Jolla, CA.

Mackie, K.R., and Stojadinovic, B. (2005). "Fragility basis for California highway overpass bridge seismic decision making." PEER Report No. 2005/02, Pacific Earthquake Engineering Research Center, University of California, Berkeley.

Mackie, K. and Stojadinovic, B. (2006). Seismic Vulnerability of Typical Multi-span California Highway Bridges, Proceedings of the Fifth National Seismic Conference on Bridges and highways, September 18-20, San Francisco.

Mackie, K.R., Wong, J-M., and Stojadinovic, B. (2008). "Integrated probabilistic performance-based evaluation of benchmark reinforced concrete bridges." PEER Report No. 2007/09, Pacific Earthquake Engineering Research Center, University of California, Berkeley.

Mackie, K.R., Wong, J-M., and Stojadinovic, B. (2010). Post-earthquake bridge repair cost and repair time estimation methodology. Earthquake Engineering and Structural Dynamics, 39(3): 281-301.

Mackie, K.R., Wong, J-M., and Stojadinovic, B. (2011). Bridge damage and loss scenarios calibrated by schematic design and cost estimation of repairs. Earthquake Spectra, 27: 1127-1145.

Maroney, B.H., and Chai, Y.H. (1994). Seismic Design and Retrofitting of Reinforced Concrete Bridges, Proceedings of 2nd International Workshop, Earthquake Commission of New Zealand, Queenstown, New Zealand.

Mander, J.B., Priestley, M.J.N., and Park, R. (1988). Theoretical Stress-Strain Model for Confined Concrete, Journal of the Structural Division ASCE, 114, pp. 1804-1826.

Mazzoni, S., McKenna, F., Scott, M. H., Fenves, G. L., et al. (2009). Open System for Earthquake Engineering Simulation, User Command-Language Manual, Pacific Earthquake Engineering Research Center, University of California, Berkeley, OpenSees version 2.0, May.

Megally, S.H., Seible, F., Bozorgzadeh, A., Restrepo, J. and Silva, P.F. (2003). Response of Sacrificial Shear Keys in Bridge Abutments to Seismic Loading, Proceedings of the FIB Symposium on Concrete Structures in Seismic Regions, May 6-9, Athens, Greece.

Shamsabadi, A., Rollins, K.M, and Kapuskar, M. (2007). Nonlinear Soil-Abutment-Bridge Structure Interaction for Seismic Performance-Based Design, Journal of Geotechnical and Geoenvironmental Engineering, 133(6), 707-720, June.

Shamsabadi, A., Khalili-Tehrani, P., Stewart, J.P., and Taciroglu, E. (2010). Validated Simulation Models for Lateral Response of Bridge Abutments with Typical Backfills, J. Bridge Eng., 15(3), 302-311, May.

Werner, S.D. (1994). Study of Caltrans' Seismic Evaluation Procedures for Short Bridges, Proceedings of the 3rd Annual Seismic Research Workshop, Sacramento, California.

Yang, Z., Elgamal, A., and Parra, E. 2003. A computational model for cyclic mobility and associated shear deformation, Journal of Geotechnical and Geoenvironmental Engineering, 129 (12), 1119-1127.

Zhang, J., and Makris, N. (2002). Kinematic Response Functions and Dynamic Stiffnesses of Bridge Embankments, Earthquake Engineering & Structural Dynamics, 31(11), pp. 1933-1966.